

# **Board Station for New Users Training Series**

## **Module 7: Creating Manufacturing Data**

Software Version 8.5\_2



Copyright © 1991-1996 Mentor Graphics Corporation. All rights reserved.  
Confidential. May be photocopied by licensed customers of  
Mentor Graphics for internal business purposes only.

The software programs described in this document are confidential and proprietary products of Mentor Graphics Corporation (Mentor Graphics) or its licensors. No part of this document may be photocopied, reproduced or translated, or transferred, disclosed or otherwise provided to third parties, without the prior written consent of Mentor Graphics.

The document is for informational and instructional purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in the written contracts between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

**RESTRICTED RIGHTS LEGEND** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in the subdivision (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013.

A complete list of trademark names appears in a separate "Trademark Information" document.

Mentor Graphics Corporation  
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

This is an unpublished work of Mentor Graphics Corporation.

TABLE OF CONTENTS

About This Module .....xv

    Workbook Organization ..... xv

    Related Documentation ..... xv

    Documentation Conventions..... xv

    Installation Procedure ..... xv

Lesson 1

    Changing References and Other Attributes ..... 1-1

        Objectives..... 1-2

        Changing Reference Designators ..... 1-2

        Changing Reference Characteristics..... 1-4

        Changing References Automatically ..... 1-5

        Lab Exercise ..... 1-7

Lab 1

    Changing References and Other Attributes ..... 1-9

        Introduction ..... 1-9

        Procedure ..... 1-10

            Preparation for Lab ..... 1-10

            Changing Reference Designations ..... 1-12

            Changing Text Appearance and Placement ..... 1-14

## TABLE OF CONTENTS [Continued]

### Lesson 2

<b>Artwork and Aperture Data .....</b>	<b>2-1</b>
Objectives.....	2-2
The Artwork Order.....	2-3
Creating an Artwork Order .....	2-4
Creating an Artwork Order for a Split Power Plane Board.....	2-8
Creating Split Power Planes.....	2-9
Creating Power Fill Areas for Split Power Planes .....	2-11
Aperture Table Functions .....	2-17
Defining an Aperture Table .....	2-18
Changing an Aperture Table .....	2-19
Displaying Power Apertures.....	2-21
Changing Artwork Format .....	2-22
Creating Artwork Data .....	2-23
Opening Artwork Data.....	2-25
Simulating Artwork Data .....	2-26
Lab Exercise .....	2-28

### Lab 2

<b>Artwork and Aperture Data .....</b>	<b>2-29</b>
Introduction .....	2-29
Procedure .....	2-30
Preparation for Lab .....	2-30
Creating the Artwork Order .....	2-30
Creating an Aperture Table .....	2-34
Defining Power Apertures.....	2-36
Specifying Power Fill Areas on Split Power Planes .....	2-37
Create -15V and Ground Power Fills .....	2-42
Creating Artwork Data .....	2-47
Simulating the Artwork Data .....	2-49

**TABLE OF CONTENTS [Continued]**

**Lesson 3**

**Creating Test Coupons and Panel Geometries ..... 3-1**

Objectives..... 3-2

Test Coupons ..... 3-3

The Panel Geometry..... 3-4

Creating a Step-and-Repeat Panel..... 3-5

Thieving Patterns..... 3-9

Creating Thieving Patterns..... 3-10

Creating Panel Artwork Data ..... 3-12

Lab Exercise ..... 3-13

**Lab 3**

**Creating Test Coupons and Panel Geometry ..... 3-15**

Introduction..... 3-15

Procedure..... 3-16

    Preparation for Lab ..... 3-16

    Creating a Link to a Geometry Library ..... 3-16

    Constructing a Panel..... 3-17

    Creating a Board Layer Block ..... 3-28

    Creating Artwork Data ..... 3-29

    Simulating Artwork Data ..... 3-32

TABLE OF CONTENTS [Continued]

Lesson 4

Creating Drilling and Milling Data..... 4-1

    Objectives..... 4-2

    Drill Table Functions ..... 4-2

    Defining and Saving the Drill Table ..... 4-3

    Restoring a Previously Saved Drill Table ..... 4-4

    Producing Drill Data ..... 4-4

    Drill Data File Naming..... 4-6

    Assigning Drill Symbols to Drill Sizes..... 4-7

    Simulating Drilling ..... 4-8

    Defining the Milling Tool Paths..... 4-9

    Changing the Milling Tool Size ..... 4-12

    Hide Milling Arrows and Order Numbers ..... 4-12

    Opening Drill/Mill Data..... 4-14

    Lab Exercise ..... 4-15

Lab 4

Creating Drilling and Milling Data..... 4-17

    Introduction ..... 4-17

    Procedure ..... 4-18

        Preparation for Lab ..... 4-18

        Creating Drill Data ..... 4-18

        Creating Milling Data ..... 4-20

## TABLE OF CONTENTS [Continued]

### Lesson 5

#### **Creating Fabrication and Assembly Drawings.....5-1**

Objectives.....	5-2
Drafting and Reports .....	5-2
Creating a Fabrication Drawing .....	5-3
Customized Drill Schedules.....	5-4
Creating an Assembly Drawing .....	5-7
Basic Drafting.....	5-7
Adding the Board's Side View.....	5-8
Adding Dimensions .....	5-8
Setting Dimensioning Style.....	5-9
Dimensioning Styles .....	5-10
Text.....	5-11
Selecting Points to Dimension.....	5-12
Adding Dimensions .....	5-13
Adding Ordinate Dimensions .....	5-15
Manufacturing Reports.....	5-16
The Aperture Table Report.....	5-16
The Drill/Mill Table.....	5-17
Customized Bill of Materials .....	5-18
Lab Exercise .....	5-19

### Lab 5

#### **Creating Fabrication and Assembly Drawings.....5-21**

Introduction .....	5-21
Procedure .....	5-22
Preparation for Lab .....	5-22
Completing a Detail.....	5-22
Creating a Fabrication Drawing.....	5-37
Creating an Assembly Drawing.....	5-46

## TABLE OF CONTENTS [Continued]

### Lesson 6

<b>Releasing a PCB Design.....</b>	<b>6-1</b>
Objectives.....	6-2
Release Methods.....	6-2
Release PCB .....	6-3
Releasing the Entire Design .....	6-3
Releasing Only PCB Data .....	6-5
Releasing Only the Schematic .....	6-6
Release PCB Summary .....	6-6
Design Configuration Concepts .....	6-8
Creating a Design Configuration.....	6-9
Opening a Configuration Window.....	6-10
Adding Primary Entries .....	6-11
Setting Build Rules .....	6-13
Building the Configuration .....	6-17
Saving the Configuration .....	6-18
Releasing a Design Configuration .....	6-19
Operations on Design Configurations and Objects .....	6-23
Viewing a Design Configuration .....	6-23
Copying a Configuration .....	6-26
Freezing a Design Object .....	6-28
Freezing a Design Configuration .....	6-30
Deleting a Design Configuration .....	6-31
Archiving and Restoring Designs .....	6-32
Lab Exercise .....	6-36



TABLE OF CONTENTS [Continued]

Lab 6

Releasing a PCB Design.....6-37

    Introduction ..... 6-37

    Procedure ..... 6-37

        Preparation for Lab ..... 6-38

        Creating a Design Configuration ..... 6-39

        Viewing the Configuration ..... 6-42

        Resetting Build Rules..... 6-43

        Creating a PCB-Only Configuration..... 6-45

        Creating a Schematic Configuration ..... 6-48

        Releasing a Design Configuration ..... 6-50

        Archiving and Restoring a Design ..... 6-54

        Removing Training Library Links from pcb\_parts Directory 6-60

## LIST OF FIGURES

PCB Design Process .....	1-1
Change Reference Popup Menu .....	1-3
Change Reference Dialog Box .....	1-4
Change All References Automatically Dialog Box .....	1-6
PCB Design Process .....	2-1
Artwork Order Example .....	2-3
Change Geometry Artwork Order Dialog Box .....	2-5
Add Artwork Layer Dialog Box .....	2-6
Highlight Icons Shown on Pins of Power Nets .....	2-12
A Completed Power Fill Polygon .....	2-14
Close-up of the Clearance Separating Two Power Fill Areas ...	2-15
Example of an Aperture Wheel .....	2-17
Add Aperture Information Dialog Box .....	2-19
Examples of Thermal Tie Graphic Options .....	2-21
Sample Artwork Format .....	2-22
Create Artwork Data Dialog Box .....	2-23
Simulated Photoplotting Output .....	2-27
Area of the +15V Pins .....	2-39
Power Fill Polygon Added Around +15V Pins .....	2-40
Close-up of Area Containing All -15V Pins .....	2-43
The -15V Power Fill Polygon .....	2-45
Close-up of Simulated Artwork .....	2-50
Close-up Showing Thermal Relief Touching Edge of Power Fill	2-51
PCB Design Process .....	3-1
Test Coupon Example .....	3-3
Close-up of a Coupon .....	3-3
Example of a Panel with Venting Pattern .....	3-4
Example of a Step-and-Repeat Panel .....	3-5
Create Manufacturing Panel Dialog Box .....	3-6
Placing a Board on a Step-and-Repeat Panel .....	3-8
Examples of Thieving Patterns .....	3-9
Create Thieving Data Dialog Box .....	3-11
Completed Outline .....	3-23
Target With Artwork Void Outline .....	3-24
Panel With Thieving Patterns .....	3-27

## LIST OF FIGURES [Continued]

PCB Design Process .....	4-1
Creating a Drill Table .....	4-5
Example Simulated Drill Path .....	4-8
Example of an Automatic Milling Path .....	4-10
Close-up of the Milling Path .....	4-11
Examples of Milling Paths .....	4-13
PCB Design Process .....	5-1
Create Drill Schedule Dialog Box .....	5-5
Drill Schedule for a Single Layer .....	5-6
Master Drill Schedule .....	5-6
Example Side View Drawing .....	5-8
Dimension Style Examples .....	5-10
Text Examples .....	5-11
Dimension Examples .....	5-13
Ordinal Dimension Examples .....	5-15
Portion of the Bill of Materials Dialog Box .....	5-18
Pointer You Will Add .....	5-25
Pointer Added to Detail .....	5-27
Feature Control Frame .....	5-27
Suggested Location for the Feature Control Frame .....	5-28
Two Feature Control Frames Placed Together .....	5-29
Completed Feature Control Frame .....	5-30
Location of the Horizontal Dimension of the Circle .....	5-31
Locating the Drill Hole Vertical Dimension .....	5-32
Locating the Dimension of the Fillet .....	5-33
Locating Two Lines of an Intersection .....	5-34
Locating the Dimension of the Chamfer .....	5-35
Completed Detail .....	5-36
Part Number Text Added To Board Outline .....	5-39
Board Outline With Vertical Dimensions .....	5-40
Board with Horizontal Dimensions and Pointer .....	5-41
Triangles Added Around Numbers in List .....	5-42
Customized Drill Schedule .....	5-45
Board Side View .....	5-48
Completed Fabrication Drawing .....	5-51

## LIST OF FIGURES [Continued]

Completed Fabrication Drawing .....	5-52
Completed Assembly Drawing .....	5-53
Completed Assembly Drawing .....	5-54
Board Process Flow Chart .....	6-1
Design Manager Configuration Window .....	6-8
Design Manager Windows Menu .....	6-10
Untitled Configuration Window .....	6-10
Add Configuration Entry Dialog Box .....	6-12
Set Build Rules Dialog Box .....	6-14
Object Information Report Window .....	6-16
Configuration Window After a Build .....	6-17
Save Configuration As Prompt Bar .....	6-18
[Configuration] Global Operations Submenu .....	6-20
Release Configuration Dialog Box .....	6-21
Released Design Directory .....	6-22
Viewing the Containment Hierarchy .....	6-24
Viewing the Primary and Secondary Hierarchy .....	6-25
Viewing Only Primary Entries .....	6-26
Copy Configuration Dialog Box .....	6-27
[Navigator] Report > Show Versions Menu .....	6-28
Sequence Versions Window and Popup Menu .....	6-29
[Configuration] Global Operations > Freeze .....	6-30
Delete Configuration Message Box .....	6-32
Change Configuration References Dialog Box .....	6-34

## LIST OF TABLES

Physical Layer Rules from LIBRARIAN.....	2-8
Split Power Plane Layers in an Artwork Order .....	2-9
Artwork Order for a Split Power Plane Board .....	2-33
Create Test Coupons Dialog Box Information .....	3-18
Example Drill Table .....	4-7
Standard Aperture Table Report.....	5-16
Aperture Table Report With X-Y Reversed.....	5-17
Example Drill Table .....	5-17
Setup Text Dialog Box Settings .....	5-23
Setup Pointer Dialog Box Settings.....	5-23
Setup Dimension Dialog Box Settings .....	5-24
Report Bill of Materials Dialog Box Entries .....	5-49



# About This Module

Welcome to *Module 7: Creating Manufacturing Data*. This module is part of the *Board Station for New Users Training Series*, which is designed to introduce PCB designers to Mentor Graphics Board Station tools. This module focuses on how to create data that is required in the PCB manufacturing process. For an overview of the tools covered in this series, refer to the “About This Training” section of *Module 1: Introduction to Board Station* in this training series.

## Workbook Organization

For an overview of *Board Station for New Users Training Series* organization and content, refer to the “Workshop Overview” section of *Module 1: Introduction to Board Station* in this training series.

## Related Documentation

For a complete listing of the manuals that make up the Mentor Graphics PCB documentation set, refer to the “Guide to the Documentation” section in the *PCB Products Overview Manual*, which describes how each manual can help you in the design process. The *PCB Products Glossary and Index* provides a glossary and index for the entire PCB manual set (referred to online as the PCB bookcase). Both of these manuals are available in INFORM.

## Documentation Conventions

For an explanation of the documentation conventions used in this workbook, refer to the “About This Training” section of *Module 1: Introduction to Board Station* in this training series.

## Installation Procedure

For complete instructions on installing data for this module, refer to “Installation Procedure” in the “About This Training” section of *Module 1: Introduction to Board Station* in this training series.

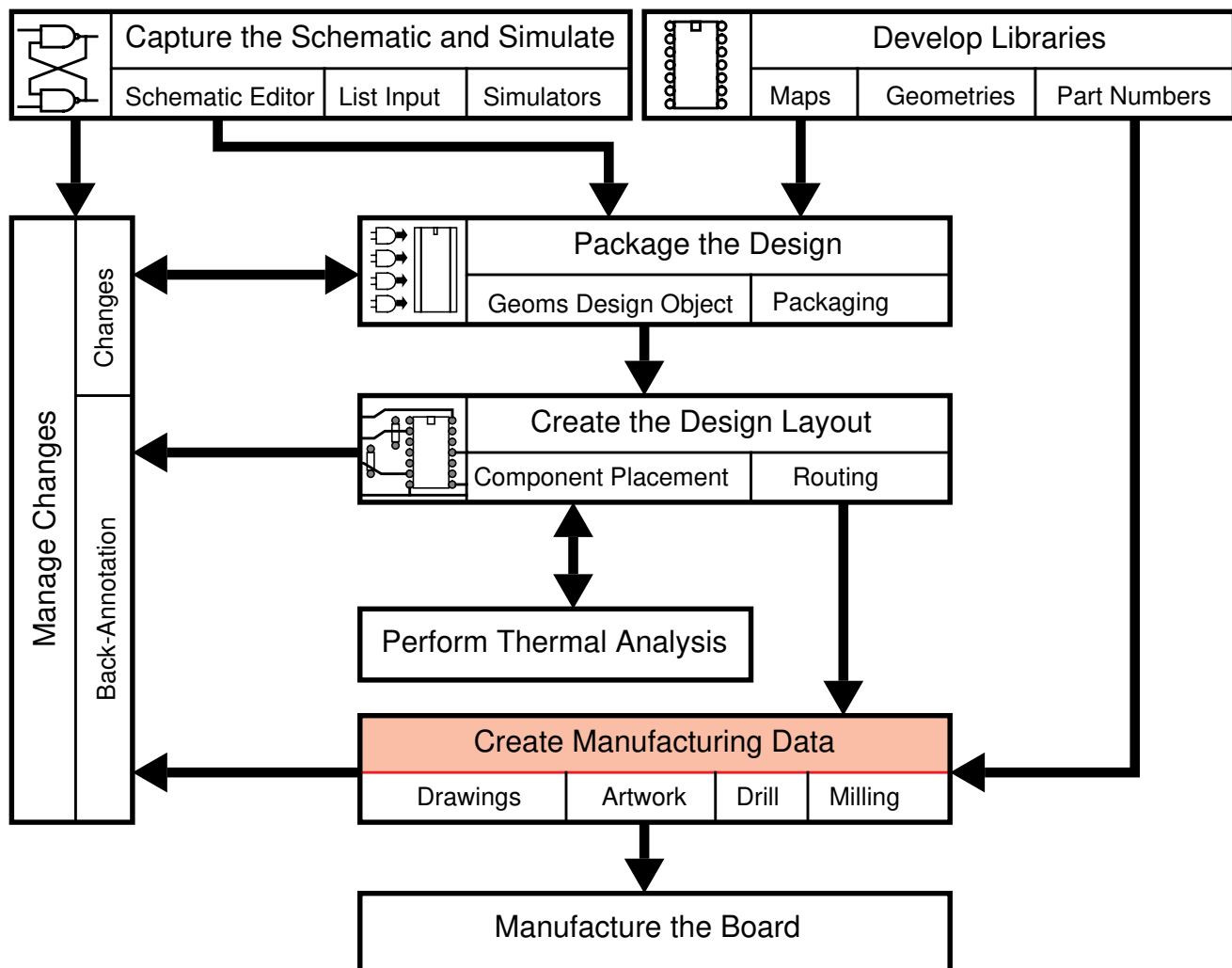




# Lesson 1

## Changing References and Other Attributes

This lesson teaches you how to place and number reference designators, as well as how to change their placement and assigned values. These steps are part of the shaded area in [Figure 1-1](#).



**Figure 1-1. PCB Design Process**

Later lessons in this module cover the process of setting up an aperture table for artwork generation and generating artwork data for a single board and for manufacturing panels. Setup and generation of drill and milling are also part of creating manufacturing data. Beyond the generation of numerical control data, manufacturing data includes drawings of the circuit board and reports such as a bill of materials.

## Objectives

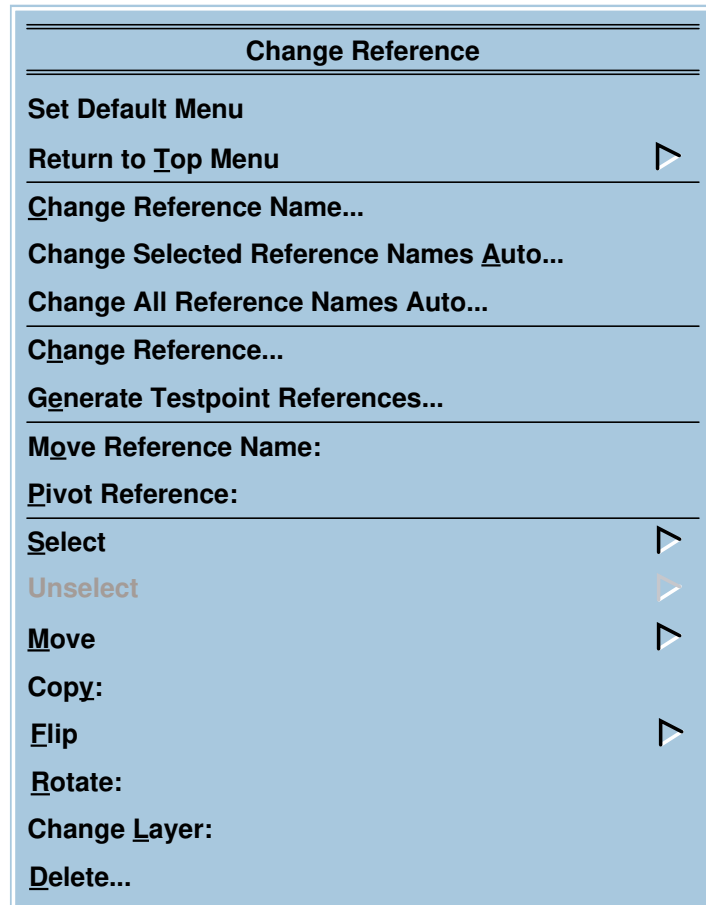
After completing this lesson, you are able to describe:

- Commands available for manipulating reference designators
- Techniques for changing reference designator characteristics

The first step in creating manufacturing data typically includes two housekeeping tasks. First, you examine the placement and numbering of reference designators. You usually adjust their placement to ensure that the designators are clearly understood, and you examine the commands used to move and pivot the designators. You also examine the command used to re-sequence reference designator numbering. Next, you examine the task of creating an Artwork Order geometry, which describes the data to plot on each sheet of film artwork.

## Changing Reference Designators

A final step in circuit board design involves the fine tuning of reference designators. Often you can improve any preliminary placement of designators made during geometry design. The **Top Menu > Change Reference** popup menu, which you access from the board geometry edit window, allows you to reposition and change the value of any reference designator. [Figure 1-2](#) illustrates the Change Reference popup menu.



**Figure 1-2. Change Reference Popup Menu**



*You can change reference designators, which are visible in a panel geometry, only while editing the board geometry.*

## Changing Reference Characteristics

To change one or more selected references on the board geometry when in FabLink, proceed as follows:

1. Place the cursor in the board geometry edit window.

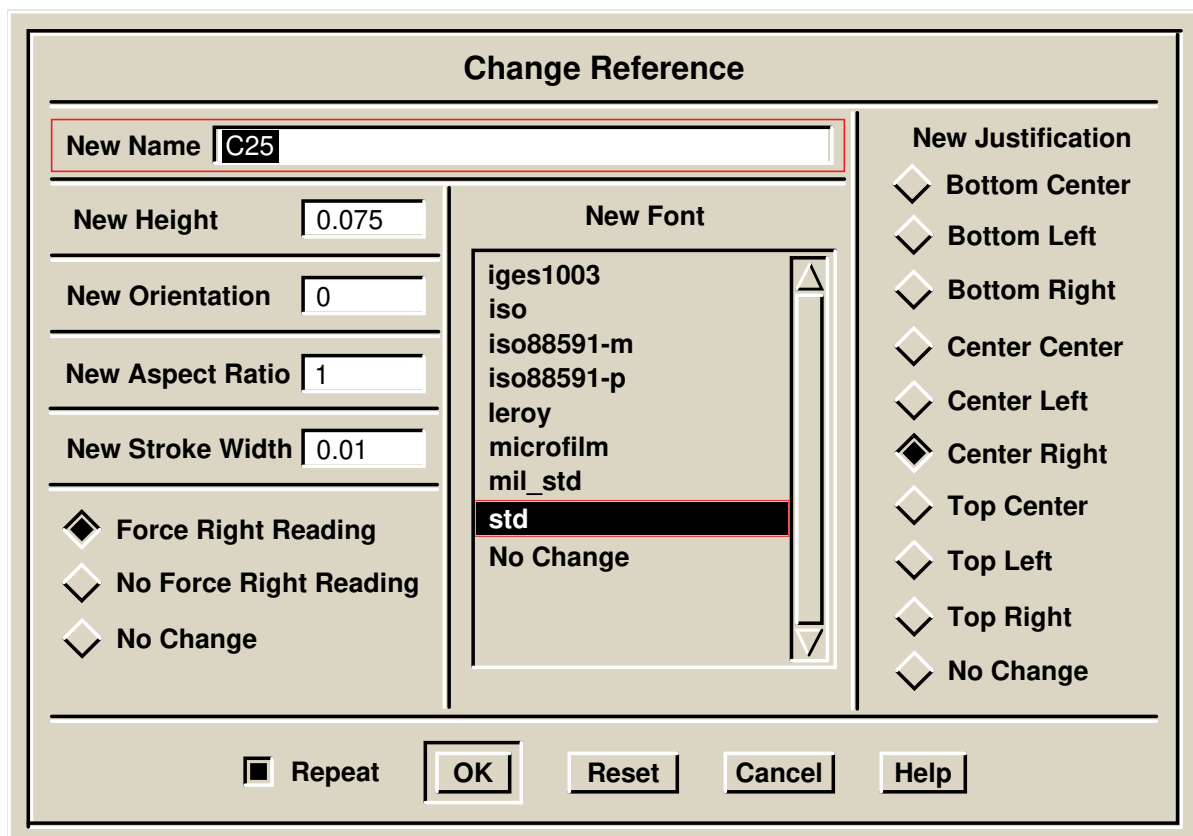
To access the menu for changing references, the board geometry edit window must be active and the silkscreen layer must be visible.

2. Choose **Top Menu > Change Reference > Change Reference**.

The Select Area prompt bar displays with the cross-hair cursor.

3. Select a reference designator to edit by placing the cursor on a designator and clicking the Select mouse button.

The Change Reference dialog box shown in [Figure 1-3](#) displays.



**Figure 1-3. Change Reference Dialog Box**

4. Make changes in the Change Reference dialog box, then press the **OK** button at the bottom of the box to execute those changes.

Upon execution, the changed reference designator replaces the original one and is then unselected.

If the **Repeat** button was selected, a Select Area prompt bar displays again so you can make another change.

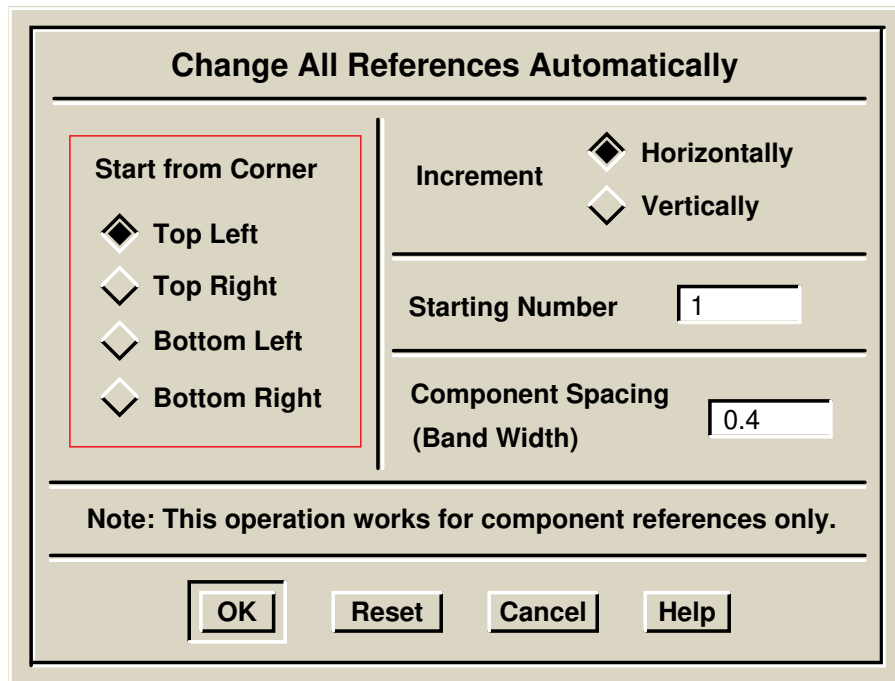
5. Either select another reference or cancel the Select Area prompt bar.

## Changing References Automatically

Selecting **Top Menu > Change Reference > Change All Reference Names Auto** in the board geometry edit window automatically sequences reference values across your board design.

You can limit the resequencing of reference values to those currently selected by choosing **Top Menu > Change Reference > Change Selected Reference Names Auto**. For example, you might want to change the reference values of only a selected group of resistors or capacitors.

Choosing **Top Menu > Change Reference > Change All Reference Names Auto** displays the dialog box shown in [Figure 1-4](#).



**Figure 1-4. Change All References Automatically Dialog Box**

The Change All References Automatically dialog box contains the following choices:

- **Start from Corner-Top Left/Top Right/Bottom Left/Bottom Right**—assigns the lowest or starting reference number in this area of the board design.
- **Increment-Horizontally/Vertically**—increases the reference number in the horizontal (row) or vertical (column) direction.
- **Starting Number**—assigns this number to the first changed reference.
- **Component Spacing (Band Width)**—numbers all components sequentially with an origin in a row or column this wide.

## Lab Exercise

In this lab exercise, you prepare your design database for the stages of artwork generation and drawing production. You review and edit the reference designators to ensure that all text has a consistent style and is located properly, and that designators are sequenced per drafting standards.

Upon completion of this exercise you can do the following:

- Select a group of reference designators.
- Change text characteristics of selected references.
- Move selected references.
- Auto-sequence the order of all reference designators.

Turn to Module 7—Lab 1: “Changing References and Other Attributes”.





# Lab 1

## Changing References and Other Attributes

### Introduction

In this lab exercise, you prepare your design database for the stages of artwork generation and drawing production. You review and edit the reference designators to ensure that all text has a consistent style and is located properly, and that designators are sequenced per drafting standards.

Upon completion of this exercise you can do the following:

- Select a group of reference designators.
- Change text characteristics of selected references.
- Move selected references.
- Auto-sequence the order of all reference designators.

## Procedure

You use FabLink to prepare your design database.

## Preparation for Lab

You can use either LAYOUT or FabLink to edit reference designators. In this lab, you use FabLink.

1. If you or your instructor have not already done so, complete the Installation Procedure in the “About This Training” section of *Module 1: Introduction to Board Station* in this training series.

2. Invoke Design Manager with the following shell command:

```
$MGC_HOME/bin/dmgr
```



3. Using Design Manager, change your current directory to the board\_new directory you just created, by clicking on the Change Directory icon in the navigator window.

4. In the Change Directory To dialog box, enter the pathname: your\_path/**training/board\_new/mod7** and press RETURN.



5. Find the FabLink icon in the Tools window, and invoke FabLink by placing the cursor on the FabLink icon and double-clicking the Select mouse button.

The INVOKING FABLINK dialog box displays.

6. In the INVOKING FABLINK dialog box, choose **sig\_az** (make certain it is in the directory *mod7* if you have done other modules of this training series). Then press the **OK** button in the dialog box.

A shell displays containing the FabLink transcript. After a moment, another shell displays that becomes the FabLink Session window.

A Report-Startup message appears in the middle of the FabLink Session window. This report is a list of notes concerning the files used to invoke the FabLink tool.

7. After reading the report notes, close the report window, and maximize the size of the FabLink session window to fill the screen.
8. Set up the view layers for editing reference designations by choosing the **View > Layers...** menu item. In the resulting dialog box, choose **All Invisible** to turn visibility of all layers off, then click on each of the following layer names so there is a V (meaning visible) next to each name.

**Board\_Outline**  
**Pad\_1**  
**Signal**  
**Silkscreen\_1**

Press **OK**. The components and their reference designators are clearly visible now.

9. Choose **Setup > Grid**. In the dialog box, specify an X increment of **0.05**, and a Display Interval of **2**, then press **OK**.

A fine grid like makes it easier for you to move and reposition reference designators later in this lab.

## Changing Reference Designations

In this procedure, you first change one reference designator, then you change the reference designators of several selected components at once, and finally you change all of the reference designators.

1. Choose **[Top Menu] Shapes > Select > Select By Name**. In the Select By Name dialog box, enter the following, then press **OK**:

Select by	<b>Property</b>
Select	<b>Components</b>
<b>From List</b>	
Properties	<b>ref</b> (scroll the list)
Value:	<b>u18</b>
Value:	<b>u19</b>
Value:	<b>u20</b>
Value:	<b>u21</b>

**OK** the dialog box, you see that four components across the upper edge of the board are selected.

2. Choose **View > Selected** to zoom in and center the view on the selected components. After the view changes, press the Unselect All function key.
3. Choose **[Top Menu] Change Reference > Change Reference Name...**, and when the Select Area prompt bar displays, click the Select mouse button on the reference designator for component U18. Complete the Change Reference Name dialog box as follows and press **OK**.

**Simple Name**  
*New Name: U19*

Reference designator U18 changes to U19, and because there was already a U19 in the design, U19 changes to U18. The system keeps track of reference designators used in the design, and makes sure only one of each is used. No other references changed.

4. Cancel the repeated Select Area prompt bar.

5. Choose **[Top Menu] Change Reference > Change Selected Reference Names Auto...**, and when the Select Area prompt bar displays, hold down the Select mouse button to define an area that completely includes the U18, U19, U20, and U21 components and their reference names. In the Change Selected References Automatically dialog box, enter the following and press **OK**.

Start from Corner **Top Left**  
Increment **Horizontally**  
*Starting number* **8**  
*Component Spacing (Band Width)* **[leave this default]**

The reference designators of the selected components change to U8, U9, U10, and U11, and the components are unselected automatically. Any components that had those reference designators is automatically renamed so that only one of each designator exists in the design.

6. Cancel the Select Area prompt bar.
7. View all of the board, then view the lower-right quarter of the board so you can see the reference designators of components in that area. Note the reference designators of several of the components; these change in the next step.
8. Choose **[Top Menu] Change Reference > Change All Reference Names Auto...**, enter the following in the Change All References Automatically dialog box, and press **OK**.

Start from Corner **Top Left**  
Increment **Horizontally**  
*Starting number* **1**  
*Component Spacing (Band Width)* **[leave this default]**

9. Choose **Yes** in the confirmation box.

The components are renamed. If not, choose **MGC > Cleanup Windows** to refresh the display.

## Changing Text Appearance and Placement

1. Change the view area so you can clearly see the reference designator of the logic components U1 and U2. Use the same procedure you used in steps 1 and 2 on page 1-12.

U1 and U2 are at the top edge of the board; they were previously named U18 and U19.

You are going to change the appearance of their reference designators, but first experiment with changing their values.

2. Choose **[Top Menu] Change Reference > Change Reference**, and when the prompt bar displays, select an area that completely includes reference designators U1 and U2 as well as their components. In the dialog box, enter *New Name: U3 U4*, and press **OK**.

No reference designator changed, and the components are unselected, because you cannot change the name of more than one reference at a time with this menu item. However, you can change the appearance of more than one component's reference designator at a time.

The Select Area prompt bar repeats.

3. Select the U1 and U2 reference designators and the associated components again. In the dialog box, specify *New Height: 0.1* and press **OK**.

The height of the reference designator text for both components is changed, and the Select Area prompt bar displays again. You can also change the value or appearance of other reference designators. However, you cannot change reference designators for the protected connectors.

4. When you are finished modifying reference designators, **Cancel** the Select Area prompt bar.

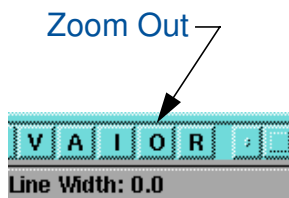
Next you practice moving reference designators.

5. Choose **[Top Menu] Change Reference > Move Reference Name**. When the Select Area prompt bar displays, click on the reference designator U1. Move the cursor to see the ghost image of the text move. Reposition the ghost image above its component and click again.

The Select Area prompt bar repeats.

6. Make sure the scroll bars are visible. If they are not, choose **Setup > Display Environment**, select **Show Scrolls** in the dialog box, and press **OK**.

Use the scroll bars in the next step to adjust the view in the edit window.



7. Select an area that completely includes reference designators U2, U3, and U4, and their components, so that all three reference designators are selected at once. To zoom out, use the zoom out icon. Move the cursor to see the ghost images of all three reference designators. Reposition them above their components, and click the Select mouse button.

8. **Cancel** the Select Area prompt bar.

Next you try pivoting a reference designator.

9. Choose **[Top Menu] Change Reference > Pivot Reference**. When the prompt bar displays, click on reference designator U2 (or any other unprotected reference designator). Move the cursor to see the ghost image pivot around the text's basepoint. Reposition the text and click again.

Instead of clicking the Select mouse button to pivot the reference, you could have entered the orientation in the prompt bar and pressed **OK**. You can also rotate a reference designation by indicating a rotation in the Change Reference dialog box.

The Select Area prompt bar does not repeat with this menu item. The reference designator is still selected.

10. Unselect all items.

11. Choose **[Top Menu] Change Reference > Pivot Reference**, and then select and pivot two or more reference designators at the same time.

Each reference designator pivots on its own basepoint. To repeat a menu item, place the cursor in the area of the session where the menu exists, hold down the SHIFT key, and click the Menu mouse button.

12. Unselect all items.

You are finished editing reference designators that are on the silkscreen.

13. Choose **File > Back Annotate**.

This menu item creates a back annotation object. Because you changed the reference designators for all of the components, you must create a back annotation object so that the new reference designator properties can be included with the design in Design Architect, where this schematic was created. Any time you modify properties in a design you must create a back annotation object so that new properties and their values can be updated in the applications where the design originated.

14. Choose **File > Save > Design All** to save all of the design. When the save is complete, **Close** the FabLink session.

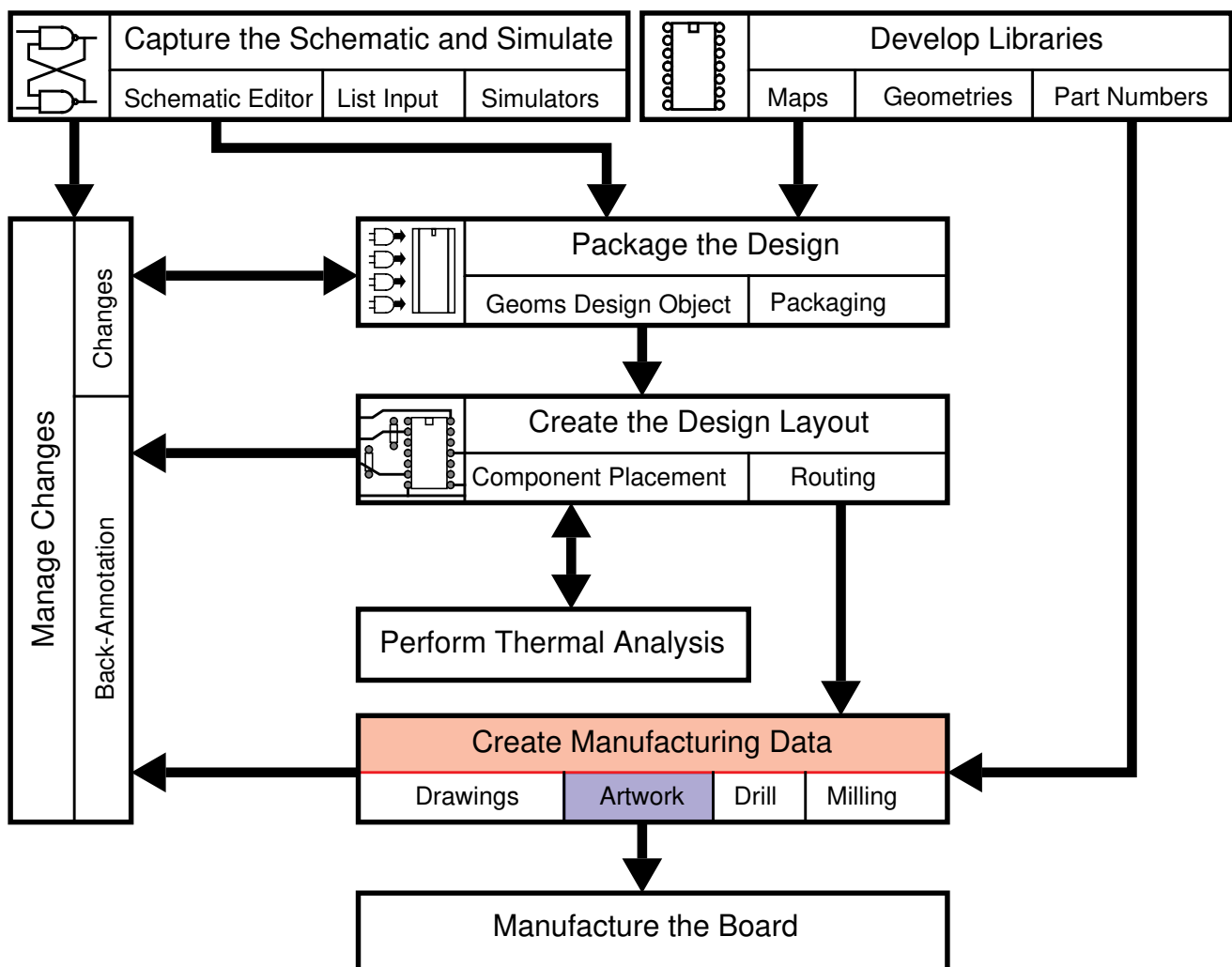
Congratulations! You have completed the “Changing References and Other Attributes” lab exercise. Continue with Lesson 2: “Artwork and Aperture Data.”



## Lesson 2

# Artwork and Aperture Data

In this lesson, you study the Artwork Order, aperture tables, split power planes, thermal reliefs, artwork data, and artwork simulations.



**Figure 2-1. PCB Design Process**

## Objectives

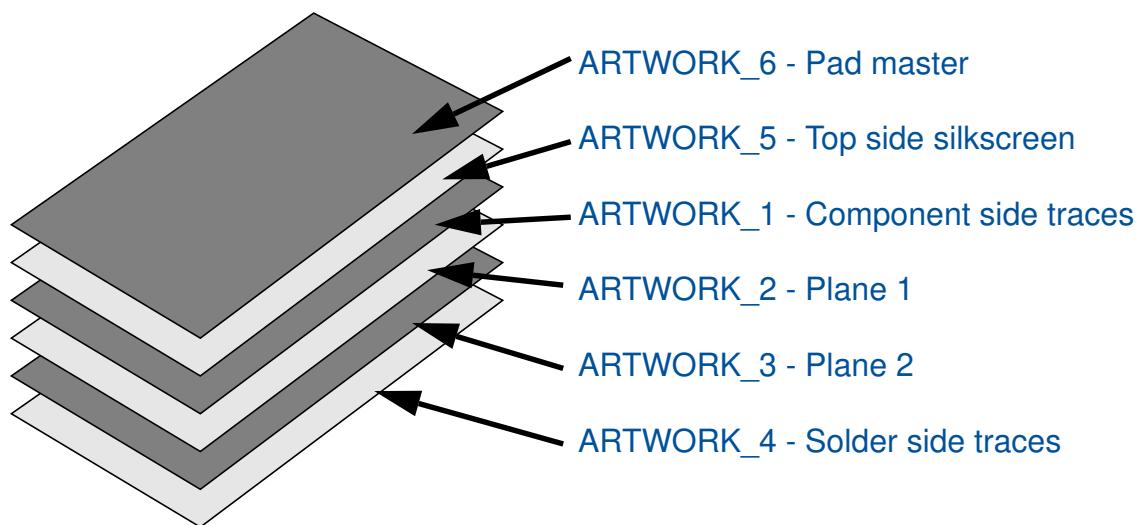
After completing this module, you are able to describe:

- The function of the Artwork Order
- How to create an Artwork Order
- The function of an aperture table
- How to create or change an aperture table
- How to create artwork for split power planes, including the use of power fill areas and highlight icons
- How to create and display thermal reliefs
- How to create artwork data
- The function and purpose of opening artwork data
- The purpose of artwork simulation

In this lesson you examine the details of creating artwork data, which is sent to a photoplotter to produce artwork film. The process requires creating an aperture table before creating artwork data. You examine aperture table creation and artwork data generation, as well as the process of simulating the artwork, to predict photoplotter behavior.

## The Artwork Order

The artwork order geometry defines which logical layers or combination of layers to produce as film for manufacturing the board. The artwork order geometry has no graphics; it consists of definitions of artwork layers. For an example, refer to [Figure 2-2](#). An artwork layer equates to a piece of film. The definition of an artwork layer assigns a logical layer or a combination of logical layers to a piece of film and sets options regarding the types of layer data to include in the artwork.



**Figure 2-2. Artwork Order Example**

FabLink uses information in the artwork order to produce design artwork. When you invoke FabLink, FabLink looks for an artwork order geometry in the *geoms* design object (a file created by PACKAGE). If FabLink does not find an artwork order geometry for the design, FabLink creates a default artwork order with the name *default\_artwork\_order*, based on the board's physical layering scheme data in the *tech* design object (a file created by PACKAGE). For more information on PACKAGE, refer to *Module 4: Packaging the Design for LAYOUT* in this training series.

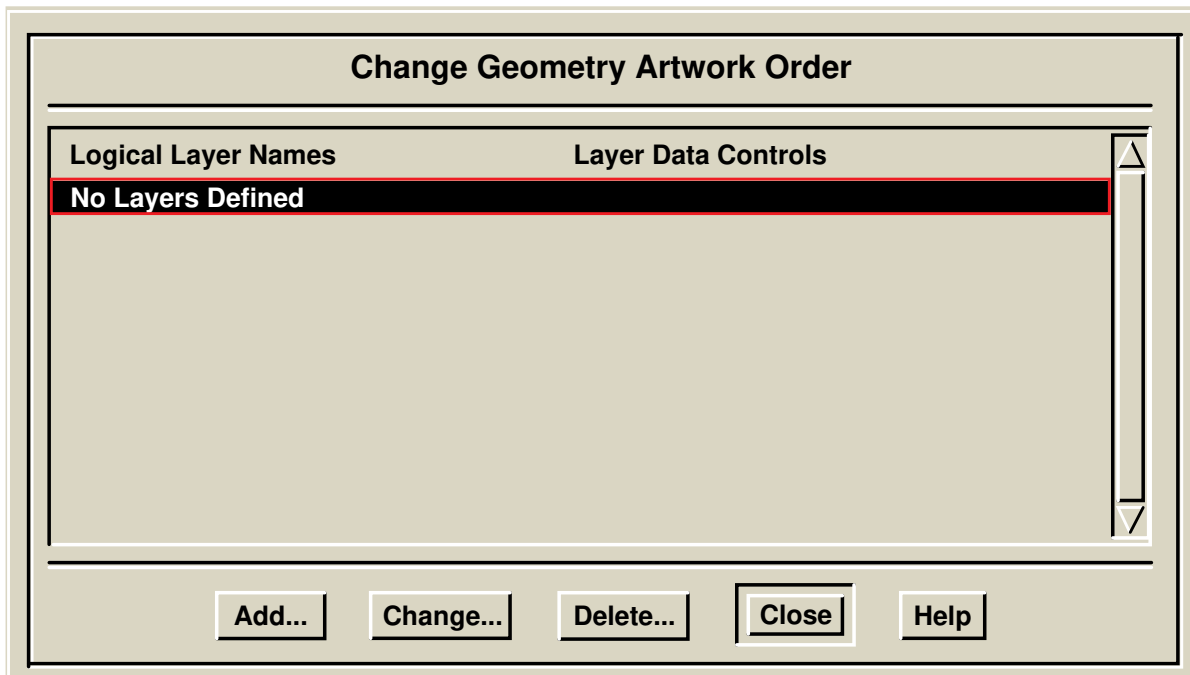
## Creating an Artwork Order

If the design already has an artwork order, such as the default, the option for creating a new one appears on the menu but is unavailable (dimmed). Because FabLink creates a default empty artwork order automatically, you must delete the default artwork order before you can create a new one to suit your needs. To delete the artwork order, choose **Geometries > Delete Geometries**.

After removing any existing artwork order, you create a new artwork order geometry by choosing **Geometries > Create Geometry > Artwork Order**.

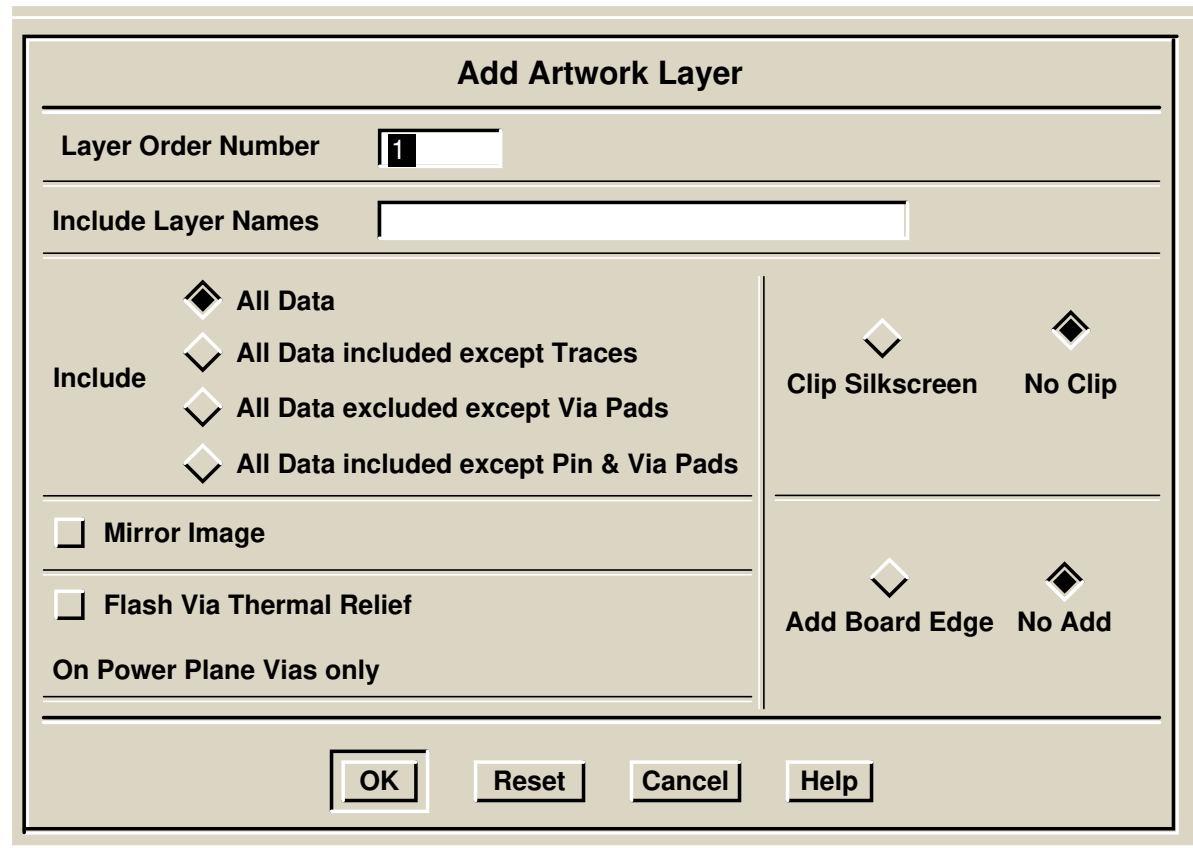
The Create Artwork Order dialog box opens and prompts you for the name of the artwork order. The entry box labeled Artwork Order Name shows the default name. To accept the default name, take no action. To assign a different name, delete the default name from the entry box and type the new name for the artwork order. Press the **OK, Add Artwork Layers** button to execute the dialog box.

FabLink creates an Edit window for the artwork order geometry and opens the Change Geometry Artwork Order dialog box, shown in [Figure 2-3](#). You can also delete and create Artwork Order Geometry in LIBRARIAN. The Change Geometry Artwork Order dialog box has two column headings: Logical Layer Names and Layer Data Controls. A highlighted line in the dialog box states: *No Layers Defined*.



**Figure 2-3. Change Geometry Artwork Order Dialog Box**

Pressing **Add** in the Change Geometry Artwork Order dialog box opens the Add Artwork Layer dialog box, shown in [Figure 2-4](#).



The dialog box is titled "Add Artwork Layer". It contains the following controls:

- Layer Order Number:** A text box containing the number "1".
- Include Layer Names:** An empty text box.
- Include:** A section with four radio button options:
  - ☒ All Data
  - ☐ All Data included except Traces
  - ☐ All Data excluded except Via Pads
  - ☐ All Data included except Pin & Via Pads
- Mirror Image:** A checkbox that is currently unchecked.
- Flash Via Thermal Relief:** A checkbox that is currently unchecked.
- On Power Plane Vias only:** A checkbox that is currently unchecked.
- Clip Silkscreen:** A radio button option that is currently selected.
- No Clip:** A radio button option that is currently unselected.
- Add Board Edge:** A radio button option that is currently selected.
- No Add:** A radio button option that is currently unselected.
- Buttons:** OK, Reset, Cancel, and Help.

**Figure 2-4. Add Artwork Layer Dialog Box**

The Add Artwork Layer dialog box provides the following controls:

- **Layer Order Number**—The entry box shows the default layer order number. To assign a different layer order number, edit the number in the box.
- **Include Layer Names**—Type the names of one or more logical layers for this sheet of film. Layers Signal\_1 and Signal\_n include through-hole pads. Layer Pad\_1 is for top-side surface mount pads; layer Pad\_2 is for bottom-side surface mount pads. To include Signal\_1 and Pad\_1 in the same artwork, assign both layers to the same artwork layer. If your design has a split power plane, enter all the power layers that are to share an artwork layer in order (for example, POWER\_1, POWER\_2, and so on).

- **Include-All Data/Pads Only/Vias Only/Traces Only**—Choose the type of data to include on the artwork. All Data includes pads, vias, tooling holes, and traces. Pads Only includes only pads, vias, and tooling holes. Vias Only includes only vias. Traces Only includes only traces.
- **Mirror Image**—This button creates a mirror image of the layer or layers on the artwork.
- **Flash Via Thermal Relief**—Press this button to create thermal reliefs on the artwork for vias that connect to power layers. If this button is not pressed, vias that connect to power layers are flooded.
- **Clip Silkscreen/No Clip**—If a silkscreen layer is associated with this artwork layer, you can automatically clip silkscreen paths that overlap pins, vias, tooling holes, text, and reference designators.
- **Add Board Edge/No Add**—A board edge is a clearance around the board outline. This clearance is useful for layers that are photoreversed in artwork, such as power or soldermask layers. For example, specify a board edge clearance if you want to flood the layer for a power net. If you choose Add Board Edge, you are prompted for a Clearance. Enter twice the clearance you require. If you want a clearance of 0.01 inch specify 0.02, because the center of the aperture for the clearance follows the edge of the board. Also, if you choose Add Board Edge, the board outline must be a path. If the board outline is a polygon, the resulting fill covers the entire board, with no cutouts.

## Creating an Artwork Order for a Split Power Plane Board

A split power plane is a single layer of a manufactured board that contains more than one power fill area. Each power fill area on the power plane connects to a different power net. For boards that have split power planes, you must specify in the Artwork Order which logical power layers are to share an artwork layer.

In LIBRARIAN, the board for the example training design was specified to have eight physical layers, as shown in [Table 2-1](#). Notice that the logical power layers are each on different physical layers in the LIBRARIAN definition.

**Table 2-1. Physical Layer Rules from LIBRARIAN**

Physical Layer	Order No.	Logical Layer 1	Logical Layer 2
Trace_layer_1	1	SIGNAL_1	PAD_1
VCC	2	POWER_1	
POS15V	3	POWER_2	
Trace_layer_2	4	SIGNAL_2	
Trace_layer_3	5	SIGNAL_3	
NEG15V	6	POWER_3	
ground	7	POWER_4	
Trace_layer_4	8	SIGNAL_4	PAD_2

To manufacture a split power plane board, you must combine the logical power layers that belong together onto a single artwork layer. The resulting Artwork Order for the same design is shown in [Table 2-2](#). [no\_thermal] is added by default.



**Table 2-2. Split Power Plane Layers in an Artwork Order**

<b>ARTWORK LAYER</b>	<b>ARTWORK FILE</b>	<b>LOGICAL LAYERS</b>
1	artwork_1	Signal_1, Pad_1
<b>2</b>	<b>artwork_2</b>	<b>Power_1, Power_2</b>
3	artwork_3	Signal_2
4	artwork_4	Signal_3
<b>5</b>	<b>artwork_5</b>	<b>Power_3, Power_4</b>
6	artwork_6	Signal_4, Pad_2
7	artwork_7	Silkscreen_1, [clip_silkscreen=0.01]
8	artwork_8	Silkscreen_2, [clip_silkscreen=0.01]

## Creating Split Power Planes

The process for creating split power planes starts at the beginning of the PCB Design Process. If you completed all of the modules in this training series, you have already completed much of this process. If not, the process up to the point of creating the power fill areas in FabLink is completed for you in your lab data. The following is a review of the entire process for creating split power planes:

1. When designing the schematic in the Schematic Editor of Design Architect, you determine which components need to share common voltages and grounds. Keep in mind that in LAYOUT, all the components with a common ground or power must be placed together (to make it easier to create a power fill that encloses all the power pins for those components). On the components you want placed together in LAYOUT, you add the placement\_region property and specify a common property value. During placement in LAYOUT, components with the same placement\_region property values are placed together in a placement area.

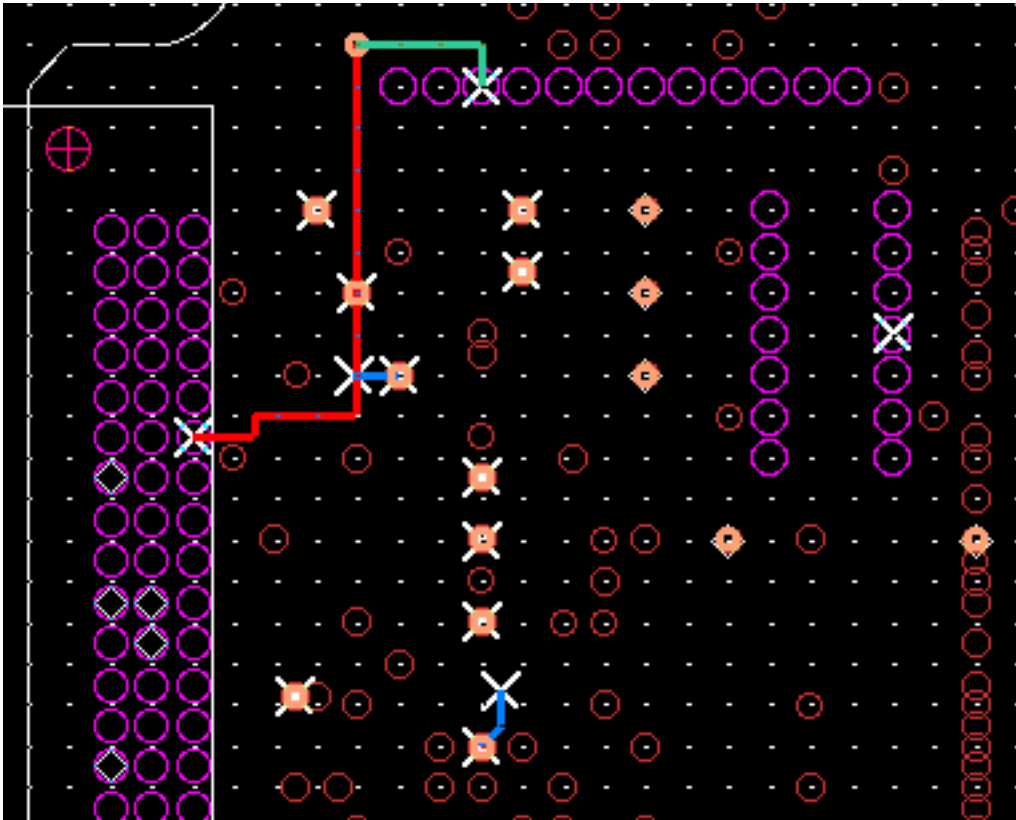
2. In LIBRARIAN, when you create the board, you specify all the Power Net Names in numerical order. The physical layers for power nets that you want on the same power plane must be contiguous, starting with the lowest power layer and ending with the highest. Refer to Lesson 4: “Creating Board Geometries” and Lab 4: “Board Geometry and Default Design Rules” in *Module 3: Creating PCB Geometries* in this training series.
3. In LIBRARIAN, when you create the board geometry, you create a placement area with the same name as the value of the Placement\_region property you assigned to components in Schematic Capture. Later, when you place components in LAYOUT, all components with the same placement\_region property value are placed in the correct placement area of the board. Refer to Lesson 4: “Creating Board Geometries” and Lab 4: “Board Geometry and Default Design Rules” in *Module 3: Creating PCB Geometries* in this training series.
4. In LIBRARIAN or FabLink, you create an artwork order layer that includes all power nets that go on the same plane. In this training series, you do this in FabLink. Refer to the section [“Creating an Artwork Order for a Split Power Plane Board”](#) on page 2-8. You can do this step either before or after LAYOUT.
5. In LAYOUT, when placing components, group together the components with the same power connections. If you group all the connections to one power net together so that they are apart from connections to different power nets, it is easier in FabLink to create a power fill area around these connections. Using a placement area on the board and the placement\_region property on the components helps with this grouping.
6. In FabLink, you create the power fill areas. The process for creating the power fill areas is discussed next.

## Creating Power Fill Areas for Split Power Planes

When you create a split power plane, you must create a power fill area for every power net on that plane (artwork layer). Create power fill areas that enclose all the pins of the power fill's connected net and that include no pins of other power nets. The process for creating power fill areas is as follows:

1. To make the view in the edit window easier to understand, limit the view to a minimum set of view layers. Typically, the set of view layers includes the Board\_outline, Power, Drill\_holes, Via\_usage, Power\_1, and Power\_2 layers. The power layers included in the view layers must include all the power layers on the split power plane. In the lab example, you create one split power plane with Power\_1 and Power\_2, and another with Power\_3 and Power\_4.
2. Set the edit layer to the lowest numbered power layer on the split power plane. For example, if Power\_3 and Power\_4 are on the split power plane, the edit layer must be Power\_3. If you do not know which power layers belong together on a power plane, refer to the Artwork Order for your design. You can view the contents of the Artwork Order by choosing **Report > Artwork Order**. Refer to [Table 2-2](#) for an example of an Artwork Order report.
3. Apply highlight icons to all the power pins, using a unique icon for each power net. With the icons applied, you can visually identify all the pins for each power net, which makes it easier to create the power fill areas around the correct pins and vias. Choose **Setup > Highlight Icons** to specify the type of icon you want to use (such as diamonds, cross-hairs, and so on), then choose **View > Highlight Net (Toggle) > Highlight Nets** to specify the net to which you want the icon applied. Be sure you apply an icon to each of the power net pins on the split power plane.

[Figure 2-5](#) shows a close-up of a board with highlight icons on the power net pins. An X icon is shown on pins of one net, and a diamond-shaped icon is shown on pins of another net.



**Figure 2-5. Highlight Icons Shown on Pins of Power Nets**

4. Determine the path of a polygon that, when a clearance (isolation path) is added around the polygon's boundary, the path electrically isolates the power pins and vias of one net from the power pins and vias of other power nets.



*Adjust the placement in LAYOUT to ensure the necessary isolation. If you make any adjustments to placement, be sure to choose the File > Back Annotate menu item afterwards. In some situations, you must create several disjoint planes. For examples of how to create a split power plane of this and other types, refer to the “Creating Split Power Planes” section in the [Using PCB Design Tools](#) manual.*

5. When you determine the necessary isolation path between the power nets, and therefore the borders of the power fill area, you add the path to the board as follows:

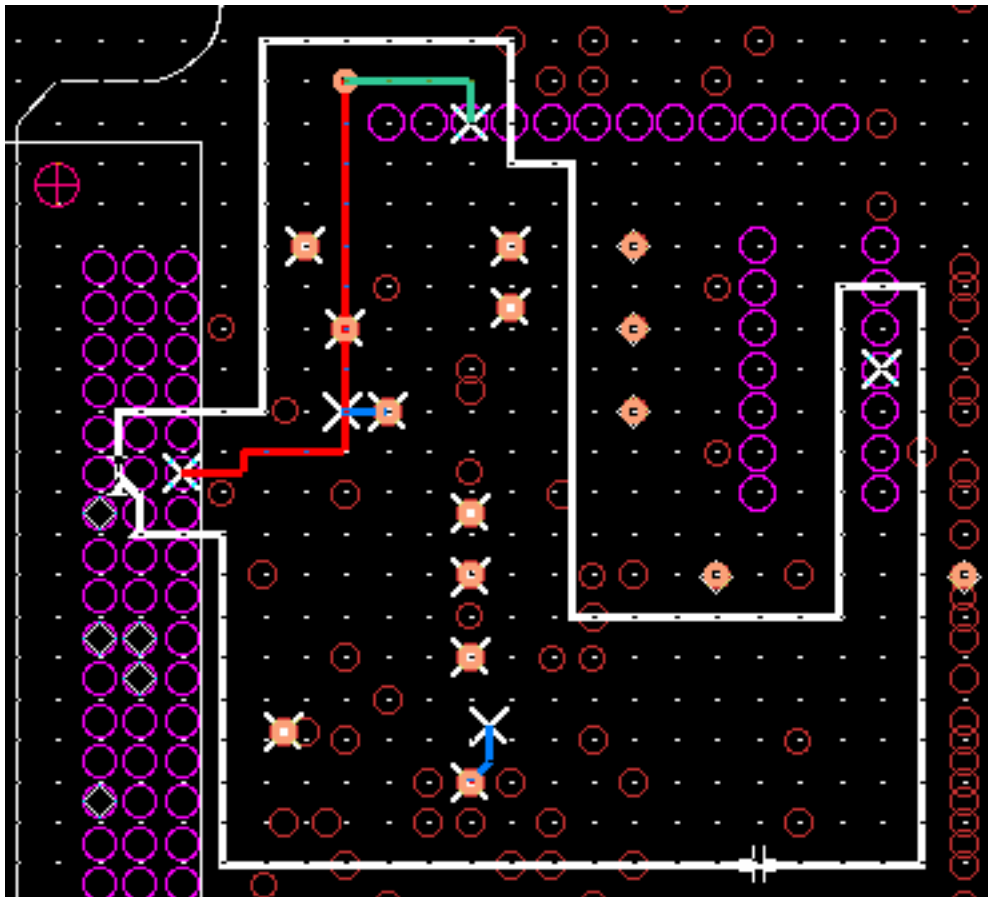
- a. Choose [**Top Menu**] **Area Fill > Power Fill**, and choose Options in the prompt bar.
- b. In the dialog box, you need to specify the Manufacturing Aperture Size. The number you enter must match a trace aperture size that exists in the aperture table (refer to section “[Aperture Table Functions](#)” on page 2-17). It is this aperture size that is used to create the power fill area. The aperture size also determines the separation between the different power fill areas.

If the size you enter is not in the aperture table, you receive a warning when you execute the **Check > Power Fills** menu item. You can specify an aperture that does not exist in the aperture table, but you must then add the required aperture to the aperture table after you create the power fill area and before you make the final check of the design.

- c. In the Add Power Fill prompt bar, press the Tab key to highlight the Net Name field and specify the name of the net to which you want the power fill area to connect. Next, Tab to the Power Layer dialog box and enter the name of the lowest numbered power layer that is on the split power plane, such as Power\_1.
- d. Tab to the Location prompt in the prompt bar. Place the cursor in the edit window where you want a power fill area polygon vertex, and click the Select mouse button. Continue clicking on vertex locations until your power fill area polygon is complete, and then either double-click on the final vertex, or choose **OK** in the prompt bar.

FabLink creates the power fill area and checks for any missing pins disconnected from the power fill area. FabLink flags disconnected pins and pins with poor coverage as warnings in a Notepad window. FabLink automatically isolates power pins of a different power net. If your power fill area is complete, the report states that all pins of the net are completely connected.

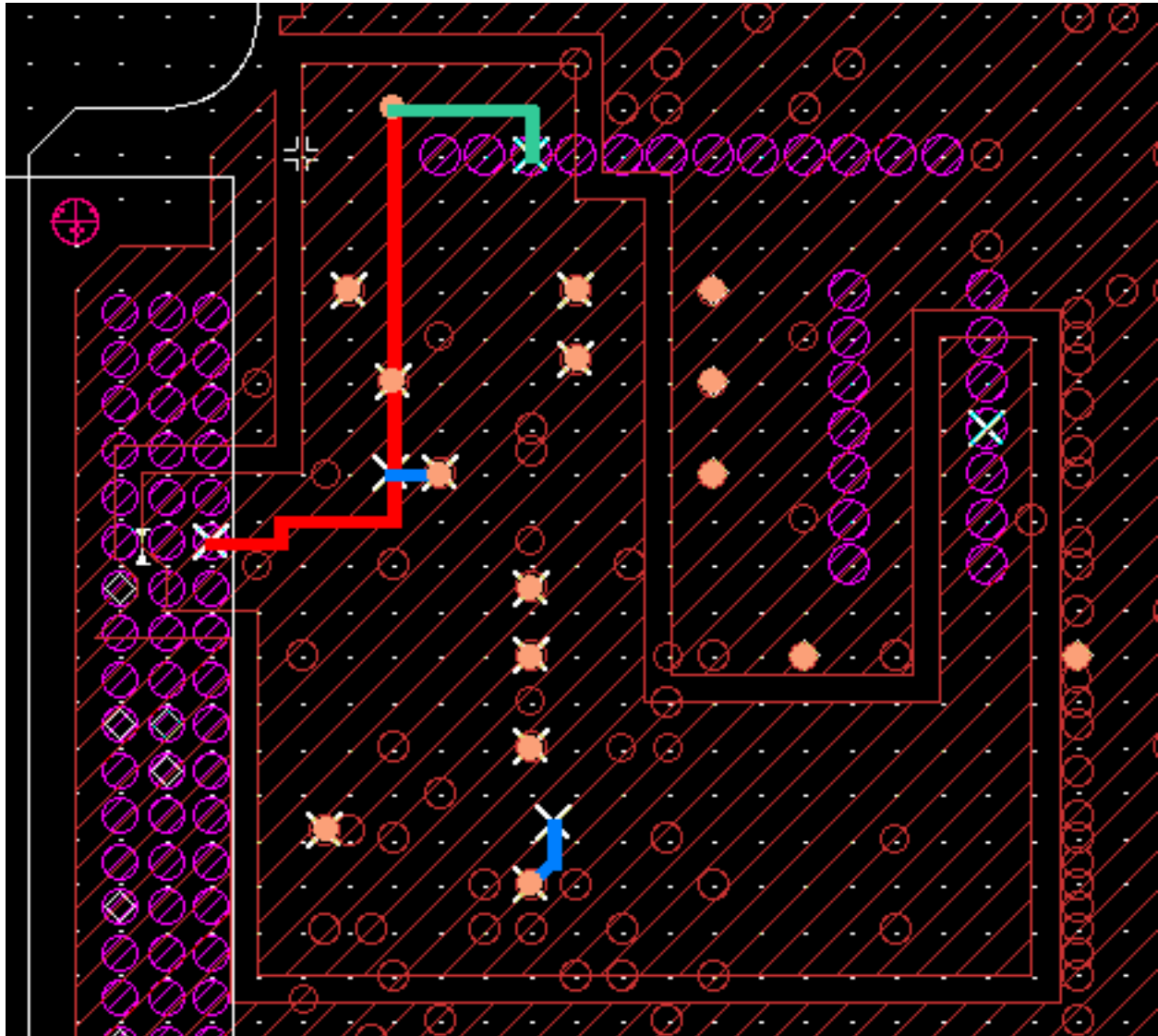
Figure 2-6 shows an example of a completed power fill polygon.



**Figure 2-6. A Completed Power Fill Polygon**

6. Repeat the previous step (adding the power fill area) for each power net on the split power plane. If two power nets are on the power plane, create two power fills to automatically create a clearance and check both power fills. Create all power fill areas on the lowest numbered power layer for that split power plane. If Power\_1 and Power\_2 are on a split power plane (physical layers Power\_1 and Power\_2 to share an artwork layer), the edit layer must be set to Power\_1 for all power fill areas. For each power fill area, you need to specify the power net name in the Net Name field of the Add Power Fill prompt bar.

Figure 2-7 shows a close-up of two power fill areas (with the view style defined so polygons are filled). You can see the clearance between the two power fill areas.



**Figure 2-7. Close-up of the Clearance Separating Two Power Fill Areas**

7. If necessary, adjust the shape of the power fill areas by creating cutouts in them. To do this, choose **[Top Menu] Area Fill > Cutout Power Fill**. First determine the net name and power layer of the power fill that you want to cut out, then specify the same net name and power layer in the Cutout Power Fill prompt bar. Cutouts created entirely within a power fill area are ignored when creating artwork, unless a different power fill is used to fill the cutout.

Power fills must cover the current edit layer entirely. Any power pins not covered by a power fill flash a thermal relief when creating artwork, with the possibility of shorts if more than one power net has pins or vias not covered by a power fill of any net.

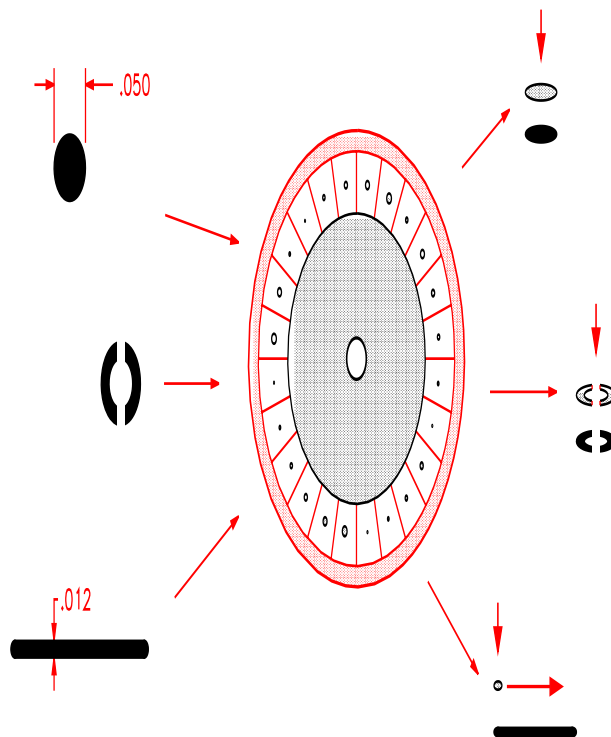
8. After you create all the necessary power fills, make a final check by choosing **Check > Power Fills**. Warnings and possible problems display in a Notepad window.
9. Save the power fill areas by choosing either **File > Save > Design > Traces** or **File > Save > Design All**.

For more information on creating split power planes, refer to the section on creating split power planes in *Using PCB Design Tools*.



## Aperture Table Functions

When you create artwork, shapes are generated in one of three ways: flashing, drawing, or painting. An aperture table correlates the pads, traces, and special symbols to the photoplotting hardware's ability to flash, draw, or paint an image. Each position on the aperture table corresponds with a position on the Gerber aperture wheel for use in the photoplotter. Refer to [Figure 2-8](#).



**Figure 2-8. Example of an Aperture Wheel**

Before creating artwork, define the aperture settings.

Your aperture table is automatically saved in your design as a design object (*design/pcb/aperture\_table*) whenever you choose either **r File > Save > Design** or **File > Save > Design > Aperture Table**, or choose **Close** from the Session window menu and then answer **Yes** in the dialog box.

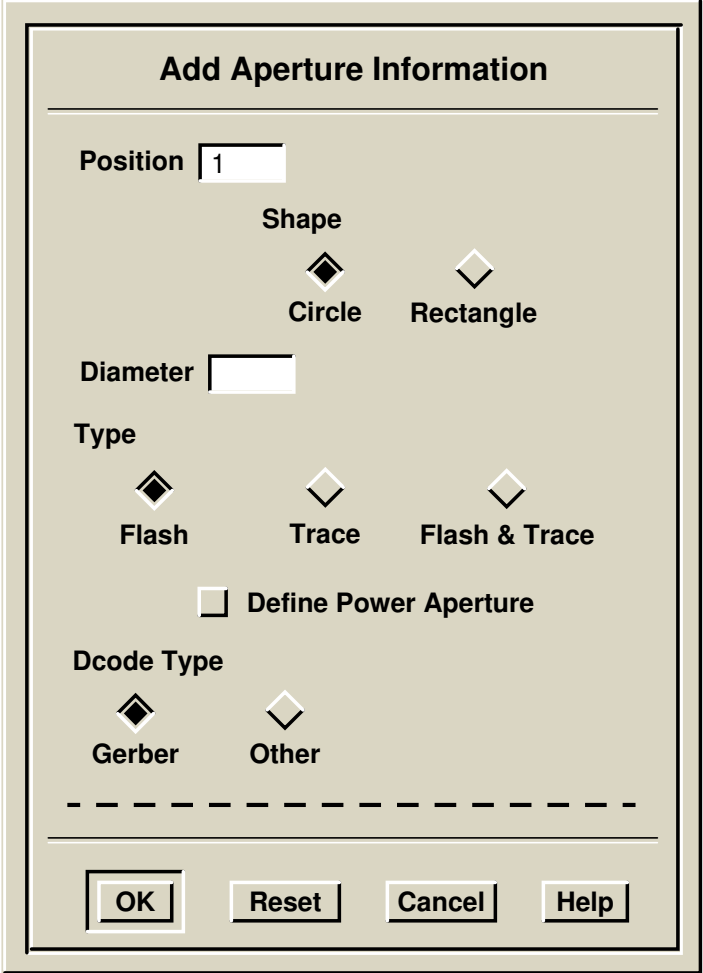
## Defining an Aperture Table

There are several ways to create an aperture table:

- If you want FabLink to automatically create an aperture table based on your design data, choose the **[Top Menu] Artwork > Change Aperture Table > Fill Aperture Table** popup menu item from the board geometry edit window. This menu item can be useful when producing artwork with a laser photoplotter, or if you want to quickly generate artwork and look at it before setting up your specific aperture table.
- If in a previous session you saved an aperture table to a custom name or pathname, you can read it back into the current session by choosing **File > Restore > Aperture Table**. In the dialog box that appears, choose the aperture table that you want to restore.
- If you need to create a new aperture position, choose the **[Top Menu] Artwork > Change Aperture Table** popup menu item from the board geometry edit window. If an aperture table is already defined in the current session, the dialog box lists current aperture table values.

## Changing an Aperture Table

To define a new aperture for the aperture table, press **Add** in the Change Aperture Table dialog box. The Add Aperture Information dialog box appears, as shown in [Figure 2-9](#).



The dialog box is titled "Add Aperture Information". It contains the following fields and options:

- Position:** A text box containing the number "1".
- Shape:** Two radio buttons labeled "Circle" and "Rectangle". The "Circle" button is selected.
- Diameter:** A text box.
- Type:** Three radio buttons labeled "Flash", "Trace", and "Flash & Trace". The "Flash" button is selected.
- Define Power Aperture:** An unchecked checkbox.
- Dcode Type:** Two radio buttons labeled "Gerber" and "Other". The "Gerber" button is selected.

At the bottom of the dialog box are four buttons: "OK", "Reset", "Cancel", and "Help".

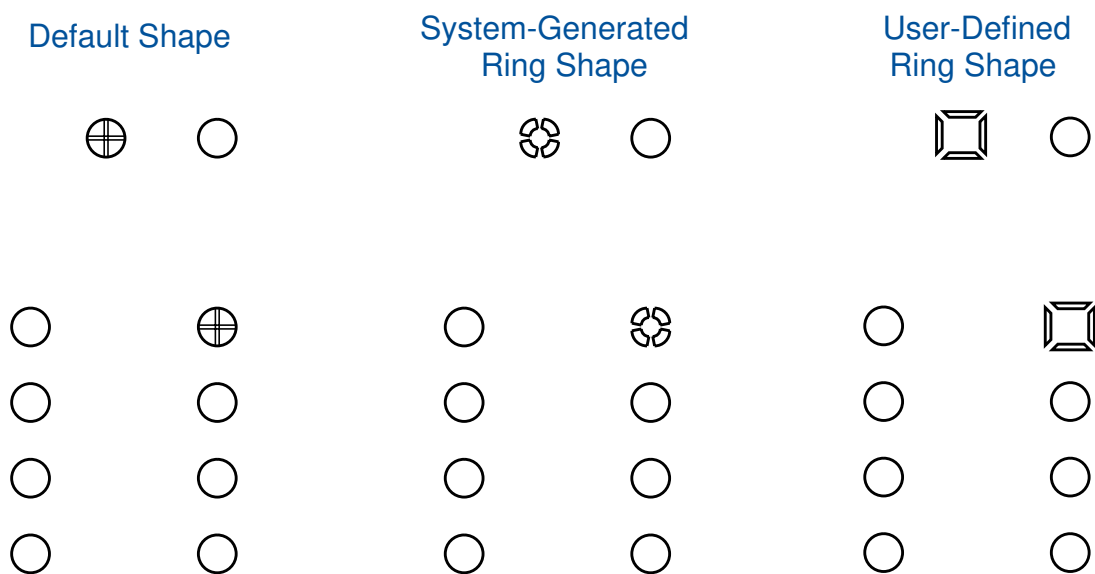
**Figure 2-9. Add Aperture Information Dialog Box**

Choices in the dialog box are:

- **Position**—type a number specifying the position in the aperture table for this aperture. Valid positions are 1-512.
- **Shape-Circle/Rectangle**—choose the shape that describes the aperture. You must supply either a diameter for the Circle choice or a width and height for the Rectangle choice.
- **Type-Flash/Trace/Flash & Trace**—choose Flash, Trace, or Flash & Trace, depending on how the aperture is used to create the image.
- **Define Power Aperture**—select the check box to mark an aperture position for creating pads that are connected to ground or voltage layers.
- **Dcode Type-Gerber/Other**—choose Gerber to select the default Dcode (drafting code) for a Gerber photoplotter. Choose Other if you want to change the default assignment of Gerber drafting codes. This option is useful if you are using a non-Gerber photoplotter, or if your Gerber photoplotter requires a different Dcode aperture assignment than the default created by FabLink.

## Displaying Power Apertures

You can mark a flash aperture position as a power aperture for creating pads that are connected to ground or voltage layers. The corresponding aperture on your photoplotter has the user-defined power aperture flash shapes for thermal ties. Because these shapes vary, there are a number of options to assist in viewing the final thermal tie shape on artwork. The thermal tie graphics options described here appear only when viewing artwork; they are not output to the artwork file(s). Examples of the options discussed in this section are shown in [Figure 2-10](#).



**Figure 2-10. Examples of Thermal Tie Graphic Options**

- You can choose the default. For this option the system displays a circle with a cross indicating that your power aperture would be flashed at this location.
- You can specify the tie width and air gap width to define a four-segment ring image. The system generates the shape automatically and substitutes the shape for the power pads.
- You can create a geometry that matches the shape used by your photoplotter to flash power apertures. Later, when you view the artwork, that shape is substituted for power pads.

## Changing Artwork Format

Before creating artwork data (Gerber artwork files), check and, if necessary, alter the format of the artwork data. The artwork format is a description of how artwork is written to artwork files. This description includes film size, data record length, use of leading/trailing zeros, and coordinate mode. Your artwork format must match the settings of your photoplotter. To check the artwork format, choose **Report > Artwork Format**. The current format, shown in [Figure 2-11](#), displays in a Notepad window.

Monday June 17, 1996; 17:01:44

### ARTWORK FORMAT

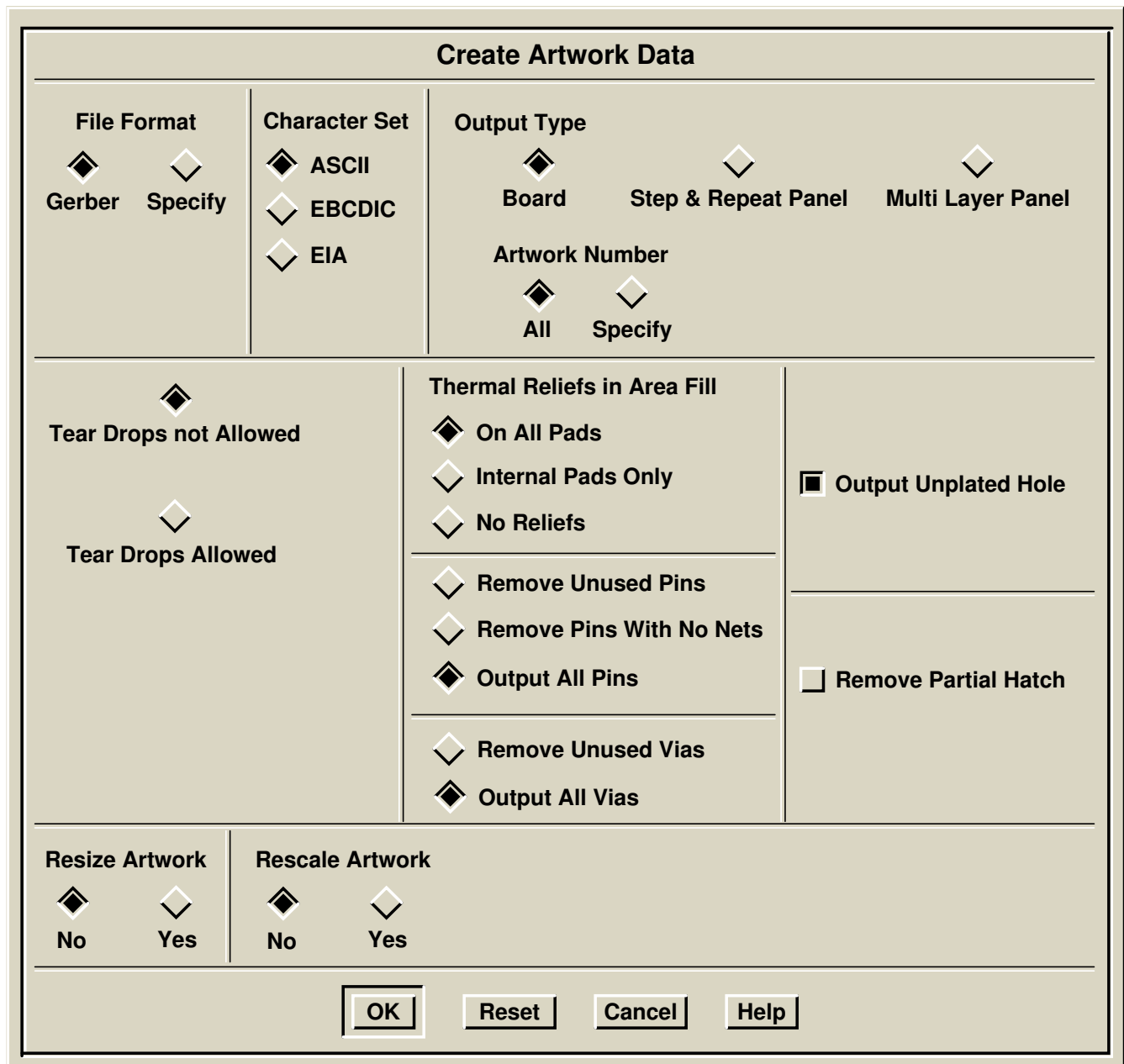
Photoplotter coordinate mode is absolute  
Photoplotter does not support circular interpolation  
Arcs and circles are interpolated by 8 segments  
Photoplotter supports Gerber G code  
Photoplotter supports inch units  
Scale is 1.000  
Data format 2.3  
Photoplotter modal coordinate is off  
Photoplotter modal shutter open(D01) is off  
Leading zeros are present  
Trailing zeros are present  
Record length of the artwork output file is 80  
Film size is (width by height): 16.000000 by 20.000000  
Automatic offset specified will center board artwork on film sheet  
Distance from film(0,0) to board origin is(x,y): 4.000000, 7.000000  
Panel artwork offset is(0,0).  
Photoplotter command block end char is '\*'  
Photoplotter machine stop code is 'M02'

**Figure 2-11. Sample Artwork Format**

It change one of these settings, such as the film size, choose the **[Top Menu] Artwork > Change Artwork Format** popup menu item, then fill out the dialog box.

## Creating Artwork Data

Generate the artwork by choosing the [Top Menu] **Artwork > Create Artwork Data** popup menu item. Options for creating the artwork appear in a dialog box, as shown in [Figure 2-12](#).



The dialog box is titled "Create Artwork Data" and contains several sections for configuring the artwork output.

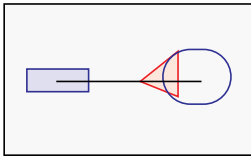
- File Format:** Radio buttons for Gerber (selected) and Specify.
- Character Set:** Radio buttons for ASCII (selected), EBCDIC, and EIA.
- Output Type:** Radio buttons for Board (selected), Step & Repeat Panel, and Multi Layer Panel.
- Artwork Number:** Radio buttons for All (selected) and Specify.
- Tear Drops:** Radio buttons for Tear Drops not Allowed (selected) and Tear Drops Allowed.
- Thermal Reliefs in Area Fill:** Radio buttons for On All Pads (selected), Internal Pads Only, and No Reliefs.
- Remove Unused Pins:** Radio buttons for Remove Unused Pins, Remove Pins With No Nets, and Output All Pins (selected).
- Remove Unused Vias:** Radio buttons for Remove Unused Vias and Output All Vias (selected).
- Output Unplated Hole:** A checked checkbox.
- Remove Partial Hatch:** An unchecked checkbox.
- Resize Artwork:** Radio buttons for No (selected) and Yes.
- Rescale Artwork:** Radio buttons for No (selected) and Yes.

At the bottom are buttons for OK, Reset, Cancel, and Help.

Figure 2-12. Create Artwork Data Dialog Box

Choices in the Create Artwork Data dialog box include:

- **File Format-Gerber/Specify**—choose Gerber unless your system is set up to handle another format.
- **Character Set-ASCII/EBCDIC/EIA**—specify the format for the character codes. ASCII is the only format you can view.
- **Output Type**—if you have created a panel, you can choose to create artwork for a board, step-and-repeat panel, or a multi-layer panel.
- **Artwork Number-All/Specify**—choose All to create artwork for every layer in your artwork order, or Specify if you want to generate specific layer sets.
- **Tear Drops**—generate teardrops on pin and/or via pads on signal layers, or on t-junctions.
- **Thermal Reliefs in Area Fill**—choose On All Pads if you want all pads inside the area fill boundary to have thermal reliefs. Choose Internal Pads Only if you want reliefs on pads that are completely inside the area fill boundary. Choose No Reliefs to flood connected pads.
- **Remove Unused Pins**—choose this option if you do not want unused pins on the artwork.
- **Remove Pins With No Nets**—choose this option to remove pins with no nets from the artwork.
- **Output All Pins**—choose this option for all pins on the artwork. This is the default.
- **Remove Unused Vias**—choose this option to remove unused vias from the artwork.
- **Output All Vias**—choose this option if you want all vias on the artwork. This is the default.





- **Output Unplated Holes**—press this check button if you want unplated through holes on the artwork.
- **Remove Partial Hatch**—remove incomplete squares of hatching from area fills that have an orthogonal hatch pattern.
- **Resize Artwork**—choose Yes to specify the absolute amount of shrinkage or growth of artwork shapes. Positive values enlarge artwork, negative values shrink artwork. The intent of this feature is to allow stepping of multiple screenings of crossover dielectric (used in hybrid technology).
- **Rescale Artwork**—choose Yes to specify a scaling factor.

## Opening Artwork Data

You can edit artwork while simultaneously viewing it by choosing the **[Top Menu] Artwork > Open Artwork Data** popup menu item. The following are general guidelines for performing the operations:

Use this method only as a last resort. Refer to the caution on [page 2-26](#).

- You can move or delete flashes. You can move, copy, delete, and create traces.
- You can add new graphical shapes. However, if you do and you want the circle to be flashed, make sure the new circle size is defined in the aperture table.
- Set the path width. Paths, arcs, and circles without a width are not plotted.
- If you use a path width that was not previously defined on the aperture table, remember to add it before saving the new artwork file. Otherwise, it is painted instead of stroked.
- Text must also have a stroke width or it is not plotted. A trace aperture with the same diameter as the text stroke width must exist in your aperture table.



*When you edit and save artwork data, only the output data is altered. The PCB design data is used to create this output (for example, your geoms design object and aperture table are not changed). You are responsible for additions and deletions; the system does not check the altered artwork data for design rule violations.*

## Simulating Artwork Data

You can simulate the photoplotting of any Gerber file, regardless of the CAD/CAM system used to generate the file. The output of the simulation displays in an Edit window, as shown in [Figure 2-13](#). The portions of the artwork created with each aperture appear on different layers. The simulation also reports the apertures used and the photoplotter head travel distance.

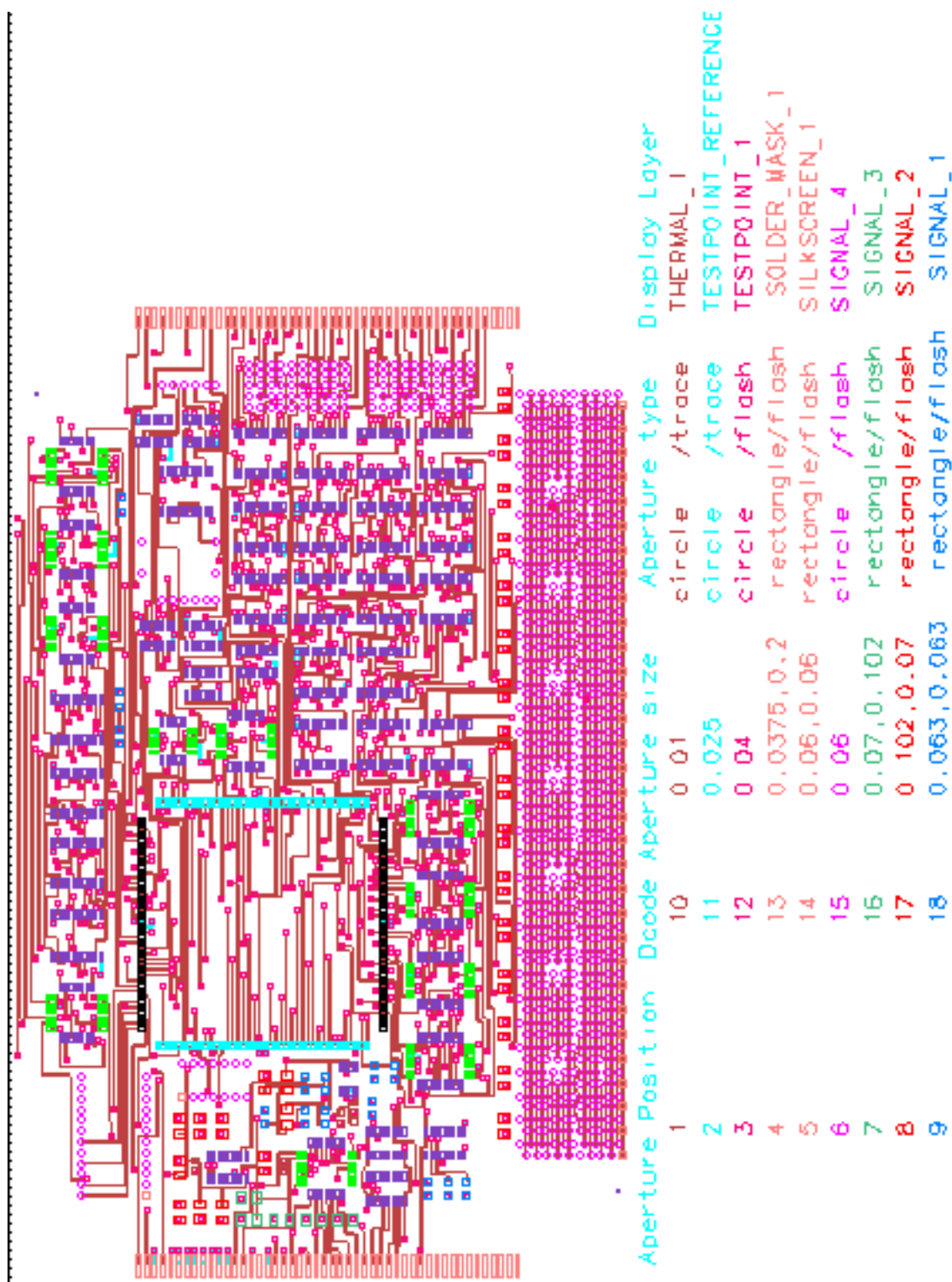


Figure 2-13. Simulated Photoplotting Output

## Lab Exercise

In this lab exercise, you perform the setup procedures for creating artwork data. You also create artwork data for several sheets of film and verify the data by simulating the artwork data.

Upon completion of this lab exercise you are able to:

- Fill an aperture table.
- Make changes to the aperture table.
- Set up the power aperture display characteristics.
- Create artwork data for the silkscreen, component side, and a power plane.
- Perform artwork simulation.

Turn to Module 7—Lab 2: “Artwork and Aperture Data”.

# Lab 2

## Artwork and Aperture Data

### Introduction

In this lab exercise, you perform the setup procedures for creating artwork data. You then create artwork data for several sheets of film and verify the data by simulating the artwork data.

Upon completion of this lab exercise you are able to:

- Create an Artwork Order and specify artwork layers.
- Fill an aperture table.
- Make changes to the aperture table.
- Set up the power aperture display characteristics.
- Add power fill areas on the power planes to isolate the +15V, VCC, -15V, and GROUND nets.
- Create artwork data for the silkscreen, component side, and a power plane.
- Perform artwork simulation.

## Procedure

In this lab exercise, you use FabLink to create artwork data.

### Preparation for Lab

1. Invoke the Design Manager, then invoke FabLink from the FabLink icon in the Design Manager Tools window. Select the **sig\_az** design in the navigator dialog box, and press **OK**.
2. Close the report window and maximize the FabLink session window.

### Creating the Artwork Order

In this procedure, you delete the `default_artwork_order` created by FabLink and create a new one with specific artwork layer definitions.

1. Choose **Geometries > Delete Geometries**. In the dialog box, choose **default\_artwork\_order**. Verify that `default_artwork_order` is highlighted and has the word *Delete* next to it. **OK** the box.

A design can have only one artwork order. You must delete the `default_artwork_order` before you can create a new artwork order.

2. Choose **Geometries > Create Geometry > Artwork Order**. In the dialog box, enter **artwork\_order** for the Artwork Order Name. Finally, choose **OK, Add Artwork Layers**.

A new edit window named `ST$artwork_order` is created; also, a dialog box displays. Next, you define the artwork order contents.

3. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number	<b>1</b>
Include Layer Names:	<b>signal_1 pad_1</b>

Layer 1 is added to the Change Geometry Artwork Order dialog box. Next, you add all the other layers to the artwork order.

4. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number:     **2**  
Include Layer Names:     **power\_1 power\_2**  
**Flash Via Thermal Relief On Power Plane Vias Only**  
**Add Board Edge**  
Clearance:                 **0.02**

This is the first split power plane. This entry makes the artwork for both power layers appear in a single artwork file. A clearance of 0.02 results in a board edge clearance of 0.01, because the center of the 0.02 trace is the edge of the board.

With this power layer, you will create a power fill for the +15V net, and you will let FabLink automatically flood the remaining area of the board up to the 0.01 board edge clearance for the VCC net. For the other power plane (-15V and ground), you will create power fills for both nets.

5. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number:     **3**  
Include Layer Names:     **signal\_2**  
Turn off **Flash Via Thermal Relief**  
**No Add** (to turn off board edge clearance)

6. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number:     **4**  
Include Layer Names:     **signal\_3**

7. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number:     **5**  
Include Layer Names:     **power\_3 power\_4**  
                          **Flash Via Thermal Relief On Power Plane Vias Only**

For this power plane you do not specify a board edge clearance because you will create the power fills for both the -15V and ground nets yourself. You need the board edge clearance only if you want FabLink to automatically flood the layer for the layer's primary net.

8. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number:     **6**  
Include Layer Names:     **signal\_4 pad\_2**  
                          Turn off **Flash Via Thermal Relief**

9. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number:     **7**  
Include Layer Names:     **silkscreen\_1**  
                          **Clip Silkscreen**  
Clearance:                **0.01**  
Include: **All Data included except Pin & Via Pads**

10. In the dialog box, choose **Add**. Enter the following information in the Add Artwork Layer dialog box. Do not change any other default entries, and **OK** only the Add Artwork Layer dialog box.

Layer Order Number:     **8**  
Include Layer Names:     **silkscreen\_2**  
                          **Clip Silkscreen**  
Clearance:                **0.01**  
Include: **All Data included except Pin & Via Pads**



11. Close the Change Geometry Artwork Order dialog box.
12. Choose **Report > Artwork Order**. Verify that the report window contents match the contents of Table 2-3. If the contents are different, change the artwork order by choosing [Top Menu] **Change This Geometry**. In the dialog box, choose **Change** and make any necessary changes.

**Table 2-3. Artwork Order for a Split Power Plane Board**

ARTWORK LAYER	ARTWORK FILE	LOGICAL LAYERS
1	artwork_1	Signal_1, Pad_1, [no_thermal]
2	artwork_2	Power_1, Power_2, [board_edge_clearance=0.02]
3	artwork_3	Signal_2, [no_thermal]
4	artwork_4	Signal_3, [no_thermal]
5	artwork_5	Power_3, Power_4
6	artwork_6	Signal_4, Pad_2, [no_thermal]
7	artwork_7	Silkscreen_1, [no_pads], [no_thermal], [clip_silkscreen=0.01]
8	artwork_8	Silkscreen_2, [no_pads], [no_thermal], [clip_silkscreen=0.01]

The signal layers are each represented on a separate sheet of film. PAD\_1 is on the film with SIGNAL\_1, and PAD\_2 shares a sheet of film with SIGNAL\_4 (the back of the board in this design). POWER\_1 and POWER\_2 share a sheet of film. POWER\_3 and POWER\_4 share a sheet of film. Later, you isolate the nets on the power layers using power fills.

13. Close the report window. Close the ST\$artwork\_order edit window.

## Creating an Aperture Table

The system generates an aperture table based on circles, rectangles, lines, and text sizes used in your design. You can also read in an existing aperture table. In this procedure, you first create the aperture definitions. You then edit an existing aperture definition.

1. Choose the **[Top Menu] Artwork > Change Aperture Table > Fill Aperture Table...** menu item. In the Fill Aperture Table dialog box, enter the following information and **OK** the box.

**Connectivity Only**

Resize Artwork: **No**

Rescale Artwork: **No**

Append Table: (This option is off/unhighlighted.)

A Report Fill\_Aperture\_Table message appears with notes about the apertures defined. The report lists all shapes that are to be photoplotted by painting, because an aperture for flashing does not exist.

You see several errors regarding reference designators for text having zero width. These errors are not important for these labs.

You might also notice some warnings stating that no clearances are defined for checking padstack geometry data against drill holes. Defining padstack drill clearances is optional. Because you did not define them in the labs, you see the warning. Do not be concerned about this warning.

Find the 0.02 inch circle listed at the bottom of the list of apertures. Notice that it is to be painted because no aperture for flashing it exists.

Close the report window.

You define the aperture for the 0.02 inch pad in the next step.

2. Choose the [Top Menu] **Artwork > Change Aperture Table > Change Aperture Table...** menu item. In the dialog box, scroll to the end of the list and note the last position number used.

You are going to add a new aperture position, so you need to know the number of the last position used.

The dialog box contains a list of all the apertures for this design. Later, when you create a power fill area on the split power planes, you specify which one of these apertures to use.

3. Choose **Add...**, enter the following in the Add Aperture Information dialog box, then **OK** the box.

Position: **[Enter a number one larger than the last number used.]**

Shape: **Circle**

Diameter: **0.02**Type: **Flash**

Define Power Aperture: (This option must be off/  
unhighlighted.)

Dcode Type: **Gerber**

You might have to scroll the list to see the new definition added at the bottom. To change an existing definition, select a position from the list and choose the **Change** option.

4. Before closing the dialog box, write down the aperture position numbers for the power apertures here (look in the Power column for *true*):

Power apertures are for thermal reliefs and power anti-pads. You need the position numbers in the next procedure when you define the power apertures for these positions.

5. Close the Change Aperture Table dialog box.

## Defining Power Apertures

There are three choices of graphical representation of the power apertures in the artwork data drawings.

- Default—power aperture display is a circle with a cross.
- Specify—thermal relief display uses a tie width and air gap width that you specify.
- Geometry—thermal relief display is a specified geometry.

In this procedure, you define the power apertures for the aperture position numbers you recorded in step 4 of the previous procedure.

1. Choose **[Top Menu] Artwork > Change Aperture Table > Change Power Aperture...**, enter the following in the Change Power Aperture dialog box, and **OK** the box.

Aperture Position: **[Enter first power aperture position number you recorded.]**

Display Power Aperture Shape: **Specify**

Tie Width: **0.020**

Air Gap Width: **0.010**

Unmark as Power Aperture: (This option must be off/unhighlighted.)

2. Repeat the previous step to create definitions for the other power apertures you recorded.

## Specifying Power Fill Areas on Split Power Planes

In this procedure, you create power fill areas for nets on the two power planes. For the power plane containing the VCC and +15V nets, you create a power fill for the +15V net, and you let FabLink automatically create artwork to flood the remainder of the power plane for the VCC net. For the power plane containing the -15V and GROUND nets, you specify a power fill area for both nets. Using the different methods for each power plane, you learn that there are several ways to create artwork for nets on split power planes.

For each power plane, the Artwork Order has combined two layers (for example, VCC and +15V) onto a single piece of artwork. You first view those layers, then view the pins, traces, and vias on those layers. After you identify the pins, traces, and vias associated with each net, you create a power fill area that contains them.

1. In the edit window containing the board, select **View > All**, so that the board fills the window. View the **Board\_outline**, **Drill\_holes**, **Power**, **Power\_1(VCC)**, **Power\_2(POS15V)**, and **Via\_Usage** layers.
2. Set the edit layer to power\_1.

You set the edit layer to the lowest numbered power layer for the set of layers on the artwork. For example, if power\_3 and power\_4 were on this artwork layer, you would set the edit layer to power\_3.



3. Choose **Setup > Highlight Icons**. In the dialog box, set Through/Blind Pin Icon to a diamond (as shown at left). Set Top Surface Pin Icon to the same kind of diamond. Set Bottom Surface Pin Icon to a diamond also. Finally, **OK** the dialog box.
4. Choose **View > Highlight Net (Toggle) > Highlight Nets**. In the dialog box, scroll to the bottom of the list of nets (all nets in the design are listed), and choose **VCC**, then **OK** the dialog box.

All pins that connect to VCC are highlighted with the diamond you specified. Next, you highlight the +15V net pins with another icon.

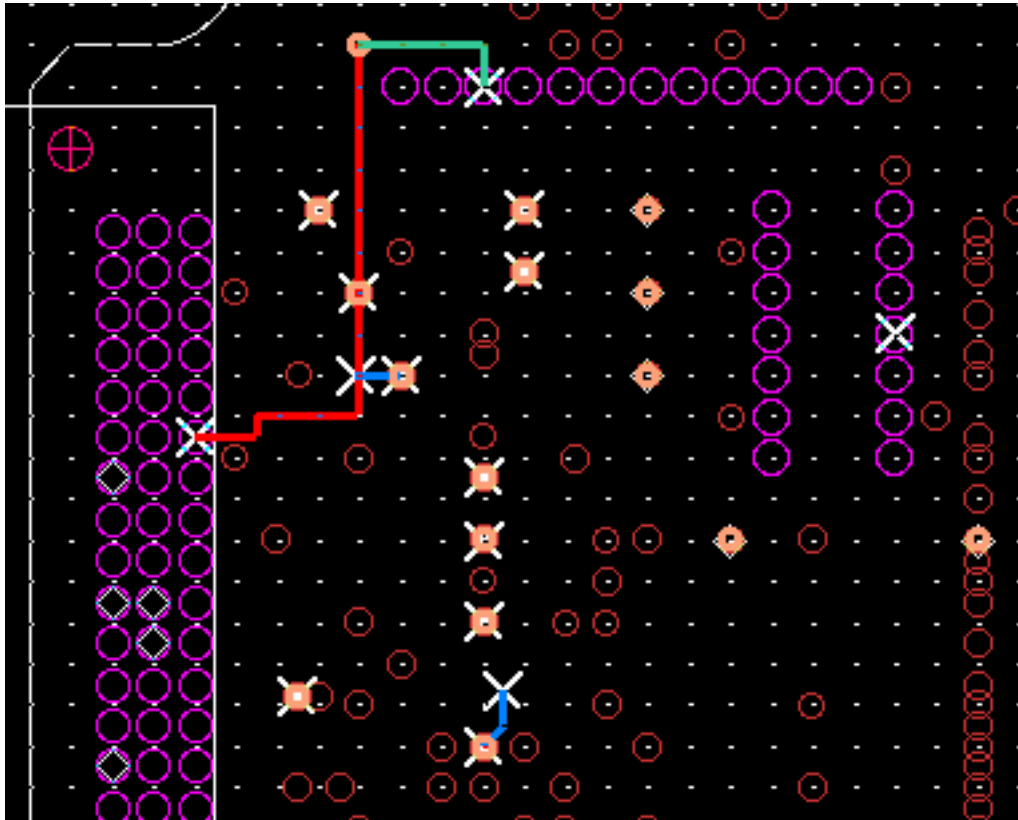


5. Choose the **Setup > Highlight Icons** menu item. In the dialog box, choose the large X icon (as shown at left) for all three pin types. **OK** the dialog box.
6. Choose **View > Highlight Net (Toggle) > Highlight Nets**. In the dialog box, scroll to the top of the list and choose +15V. **OK** the dialog box.

Now both of the nets on the power\_1 artwork layer are separately highlighted. You can now easily see that all the +15V pins are in the upper-left corner of the board. Next, you create a power fill area that includes all the +15V pins and none of the VCC pins.

7. Set the grid to an X increment of 0.05, with a display interval of 2.

8. View the area around the +15V icons (with the X highlights), as shown in Figure 2-14.

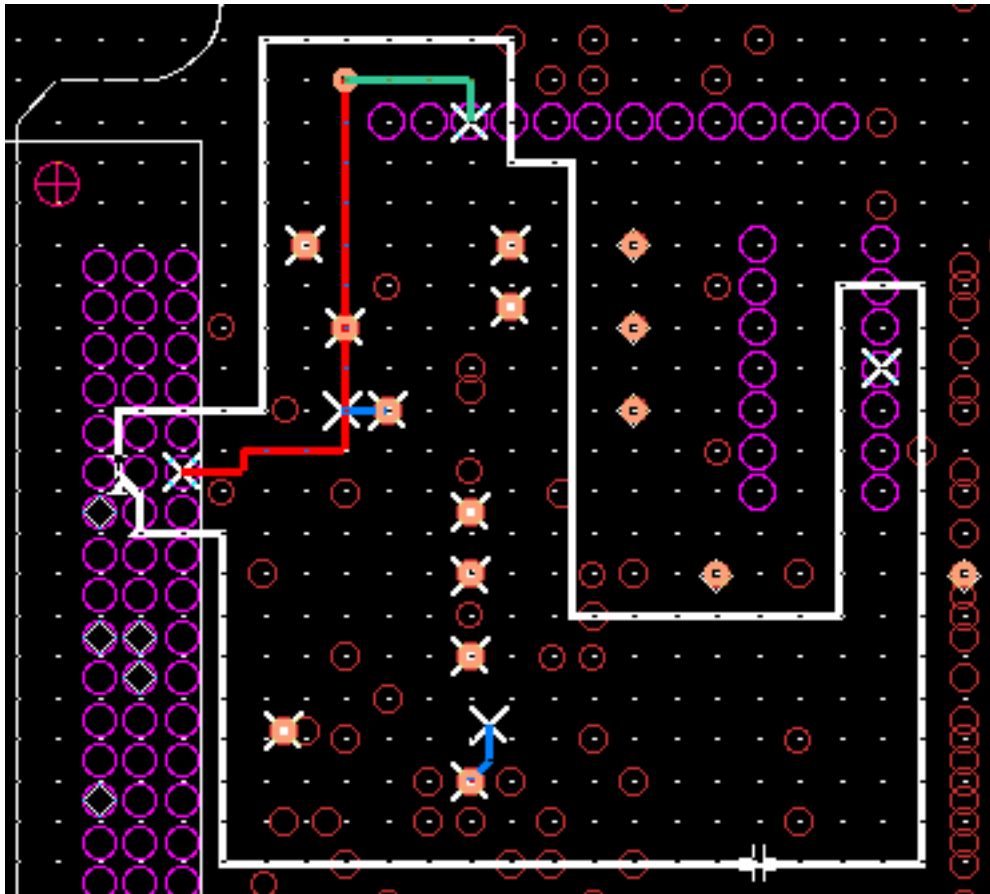


**Figure 2-14. Area of the +15V Pins**

9. Choose [Top Menu] **Area Fill > Add Power Fill**. In the prompt bar, choose **Options**. Enter **0.06** for the Manufacturing Aperture Size. **OK** the Options dialog box. In the prompt bar, enter **+15V** for the Net Name, and **POWER\_1** for the Power Layer. Tab to the locations prompt.

In the next step, imagine a polygon that encloses an area that contains only +15V pins (with the X), and does not contain any VCC pins (with the diamond icons). You click the Select mouse button on each vertex of this polygon to define the polygon.

10. Click on each vertex to form a power fill polygon. Create a polygon similar to the one shown in Figure 2-15. Be certain that there are no VCC pins (with the diamond) within the polygon you create. Continue clicking on each vertex until the polygon is closed, then double-click on the last vertex of the polygon to complete the power fill area.



**Figure 2-15. Power Fill Polygon Added Around +15V Pins**

You see a report stating that all +15V pins are connected, and the Add Power Fill prompt bar repeats.





*If you place a vertex at the wrong location, press the Backspace key to remove one vertex for each Backspace keystroke. Also, if you complete the polygon but receive an error or warning, you can use [Top Menu] Area Fill > Select > Select Fills to select the power fill polygon; you can then delete it and recreate it.*

If you place the polygon so that it passes within the Manufacturing Aperture Size (0.06 in this example) of a VCC pin, the polygon is automatically adjusted to keep it the minimum distance (0.06) from the VCC pin. In [Figure 2-15](#), there is a VCC pin near the extreme left edge of the power fill area. When the example was created, the polygon passed too close to the VCC pin, and you can see how the polygon was adjusted to keep it away. If a VCC pin is contained within the polygon, an area around the VCC pin is automatically cut out of the power fill area; however, the pin is isolated.

11. Close the report window. Close the repeated prompt bar.

You specified a power fill area for the +15V net. When you created the geometry for this board, you specified the power nets in sequential order: VCC, +15V, -15V, and GROUND. The first two nets (VCC and +15V) are on the first split power plane, and -15V and GROUND are on the other split power plane.

In the following steps, you create an area fill on the VCC layer. A fill over this entire layer guarantees that checking flags any pin that falls outside the +15V fill. Adding the fill over the entire VCC layer creates a clearance area around the edge of the board, ensuring that the copper does not run to the edge of the board.

12. View all of the board.
13. Choose [Top Menu] Area Fill > Add Power Fill, choose Options, then enter **.01** for the Manufacturing Aperture Size. **OK** the options dialog box. In the prompt bar, enter **VCC** for the Net Name, enter **POWER\_1** for the Power Layer, and Tab to the location prompt.
14. Create a power fill polygon that goes around the perimeter of the board, about 1 or 2 visible grid spaces inside the edge of the board.

## Create -15V and Ground Power Fills

In the next several steps, you add power fill areas for the -15V net and for the GROUND net.

1. View all of the board and view only the following layers: Board\_outline, Power, Drill\_holes, Via\_usage, Power\_3, and Power\_4. Make sure visibility of layers Power\_1 and Power\_2 is turned off.

2. Set up the edit layer to Power\_3.

Always set the edit layer to the lowest numbered power layer when adding power fills on split power planes.

3. Choose **View > Highlight Net (Toggle) > Unhighlight All Nets** to turn off all net icons.

This clears the display of old icons so that you can clearly see new icons for the -15V and GROUND nets.



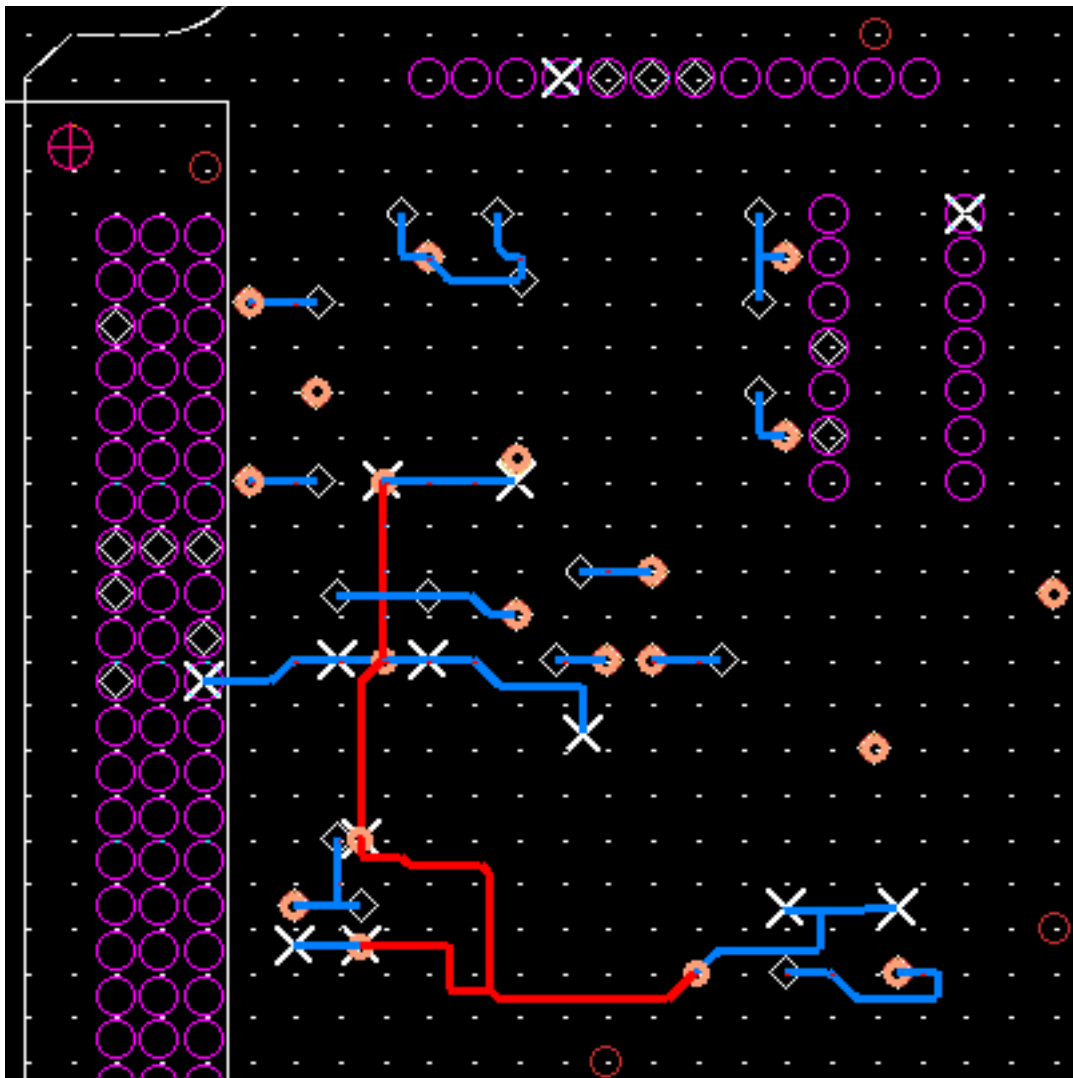
4. Choose **Setup > Highlight Icons**. In the dialog box, choose the large X icon (shown at left) for all three pin types. **OK** the dialog box.

5. Choose **View > Highlight Net (Toggle) > Highlight Nets**. In the dialog box, choose **-15V**. **OK** the dialog box.

The X icon is applied to all of the -15V pins; these are all in the upper-left corner of the board. Next, you highlight all the GROUND pins.



6. Choose **Setup > Highlight Icons**. In the dialog box, choose the sixth icon from the left (shown at left) for all three pin types. **OK** the dialog box.
7. Choose **View > Highlight Net (Toggle) > Highlight Nets**. In the dialog box, choose **GROUND** (near the bottom of the list). **OK** the dialog box.
8. View the area shown in [Figure 2-16](#).



**Figure 2-16. Close-up of Area Containing All -15V Pins**

In the next steps, you create a power fill polygon that encloses all the X icons and none of the diamond icons. Because the pins on this plane are much more densely packed than on the other power plane, use a finer grid and a smaller Manufacturing Aperture Size.

9. Set the grid to an X increment of 0.01, with a display interval of 10.

This keeps the same visible grid, with 10 snapping spaces between the visible grid.

10. Choose **[Top Menu] Area Fill > Add Power Fill**, choose Options, then enter **.01** for the Manufacturing Aperture Size. **OK** the options dialog box. In the prompt bar, enter **-15V** for the Net Name, enter **POWER\_3** for the Power Layer, and Tab to the location prompt.

Create this power fill polygon much closer to the pins because of the smaller spacing between pins on this plane.

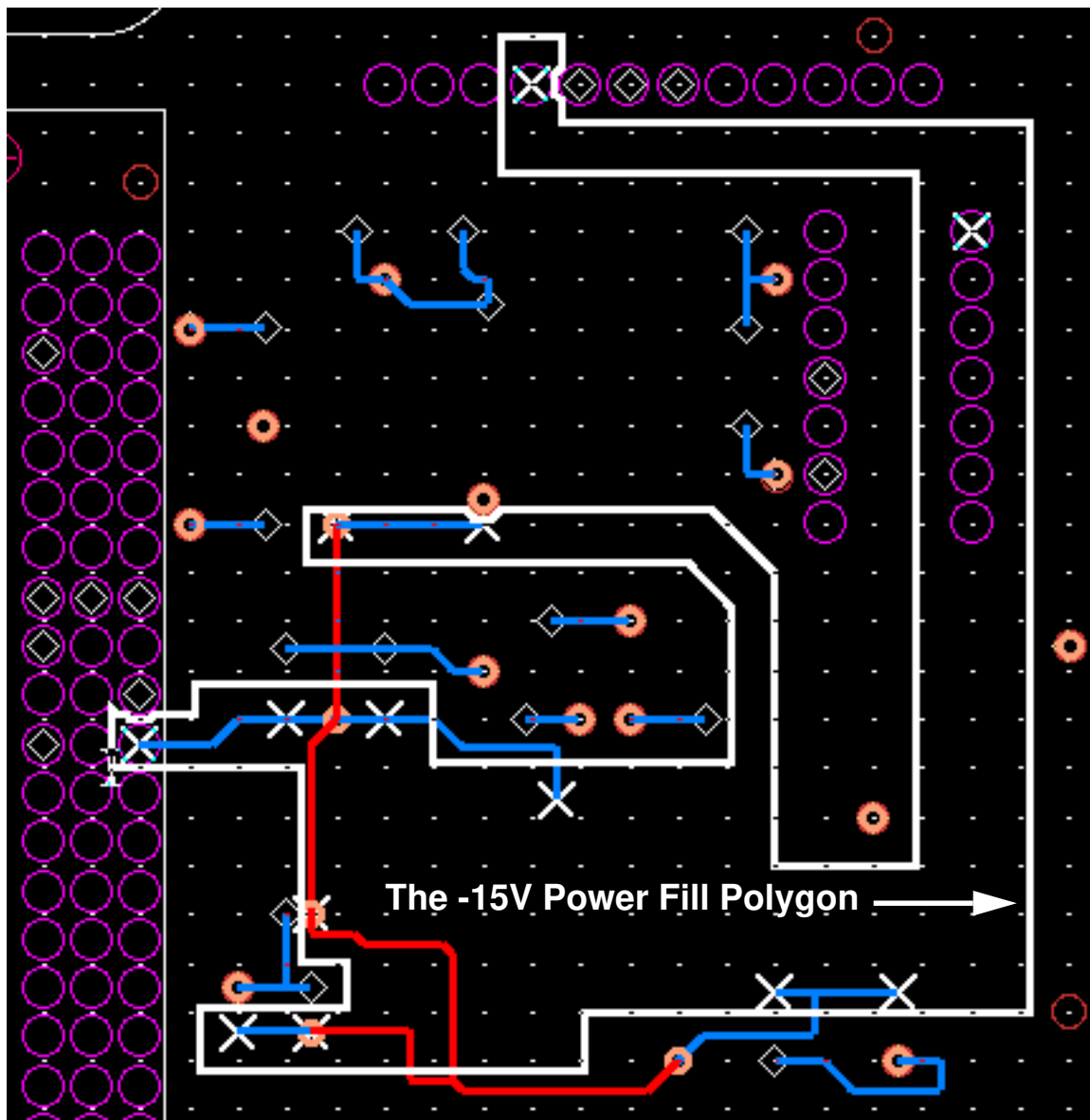
11. Create a power fill polygon like the one shown in [Figure 2-17](#). Be sure to double-click on the final vertex to close and complete the polygon.

As when you created the +15V power fill, you can draw the polygon so that it goes across the top of GROUND pins (those with the diamond icon). When the polygon is finished, FabLink redraws the power fill area around those pins. However, avoid going across GROUND pins; the more room you leave around them, the more likely they are to have good coverage when you add the GROUND power fill. Try to leave at least a 0.01 inch clearance around them to allow for the Manufacturing Aperture Size.

You see a report stating that all -15V pins are connected. The Add Power Fill prompt bar repeats. If the report states that some pins have not been connected, use **[Top Menu] Area Fill > Select > Select Fills** to select the power fill polygon, so that you can delete the power fill and recreate it.



*If you misplace a vertex, you can press the Backspace key to remove it. You can remove one vertex for each Backspace keystroke. Also, if you complete the polygon but receive an error or warning, you can use **[Top Menu] Area Fill > Select > Select Fills** to select the power fill polygon. You can then delete it and recreate it.*



**Figure 2-17. The -15V Power Fill Polygon**

12. View the entire board. In the repeated Add Power Fill prompt bar, change the Net Name to GROUND. Make sure the Power Layer is POWER\_3, and the Manufacturing Aperture Size is still 0.01. Tab to the location prompt. Create a power fill polygon around the perimeter of the board, about 1 or 2 visible grid spaces inside the edge of the board.

When you complete the power fill area for the GROUND net, you might receive some warnings about poor coverage for pins. These warnings are caused by the close spacing between the GROUND pins and the -15V pins. Because you cannot use a smaller Manufacturing Aperture Size to gain more room between the two power fill areas, the best solution is probably to open the design again in LAYOUT, and reposition the components to provide more room between the GROUND and -15V pins. Due to time limitations, you do not do that in this training.

When you simulate the artwork, you see that the poor coverage is because some of the thermal reliefs near the edge of the GROUND power fill just touch the edge of the power fill area.

**13.** Close the report window

## Creating Artwork Data

Before generating the artwork data, verify that all aperture definitions are created for the design.

1. Choose **Check > Aperture Table...**, verify the following settings in the dialog box, and press **OK**.

Check Artwork: **All Layers**  
Resize Artwork: **No**  
Rescale Artwork: **No**

A Report-Check\_Artwork message appears with notes about defined apertures. The list of painted sizes no longer includes the 0.02 circle, because you defined the circle.

2. Scroll to the bottom of the report window and locate the error about the missing 0.06 inch aperture.

When you created the first split power plane, on power\_1 for the +15V and VCC voltages, you specified a Manufacturing Aperture Size of 0.06. Because there is only a 0.06 flash aperture, but no 0.06 trace aperture in the aperture table, you now get an error. In the next step, you add the 0.06 trace aperture to the aperture table to remove the error.

3. Close the report window.
4. Choose **[Top Menu] Artwork > Change Aperture Table > Change Aperture Table**. Notice that no aperture for a circle with a type of trace and a height of 0.06 inches exists. Note the number of the last aperture position for the next step.
5. Choose **Add...**, enter the following the Add Aperture Information dialog box, and **OK** the box.

Position: **[Enter a number one larger than the last number used.]**  
Shape: **Circle**  
Diameter: **0.06**  
Type: **Trace**  
Dcode Type: **Gerber**

6. Close the Change Aperture Table dialog box.
7. Choose **Check > Aperture Table...** again. Verify that no apertures are missing. Ignore errors and warnings about zero-width reference designators and the silkscreen for reference designators overlapping pin shapes.
8. Close the report window.
9. Create the artwork data by choosing **[Top Menu] Artwork > Create Artwork Data...**, enter the following in the Create Artwork Data dialog box, and **OK** the box.

File Format:       **Gerber**  
Character Set:     **ASCII**  
Output Type:       **Board**  
Artwork Number: **All**

**Tear Drops not Allowed**  
Thermal Reliefs in Area Fill: **On All Pads**  
**Output All Pins**  
**Output All Vias**  
**Output Unplated Hole**

Resize Artwork:   **No**  
Rescale Artwork:  **No**

A Report Generate Artwork message appears with notes about the artwork generation. You can ignore any of the warnings.

10. Read and close the report window.



## Simulating the Artwork Data

The final step in verifying artwork to be generated is to run the artwork simulation. This technique displays an artwork drawing showing the actual artwork data; it also provides artwork statistics at the bottom of the drawing.

1. Choose **[Top Menu] Artwork > Simulate Artwork Data...**, enter the following in the dialog box, and press **OK**.

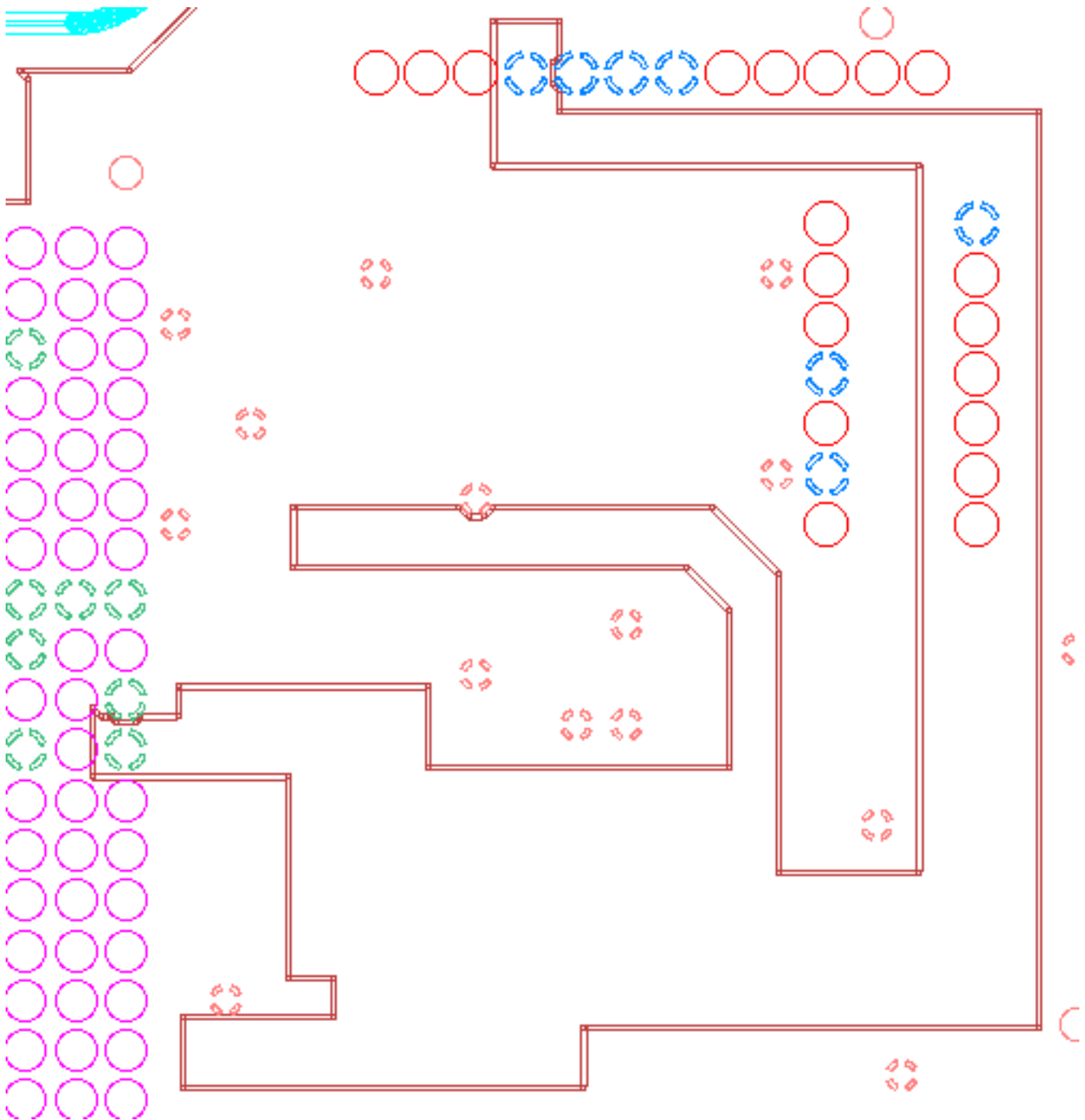
**Select**

Artwork File Names: **artwork\_5**

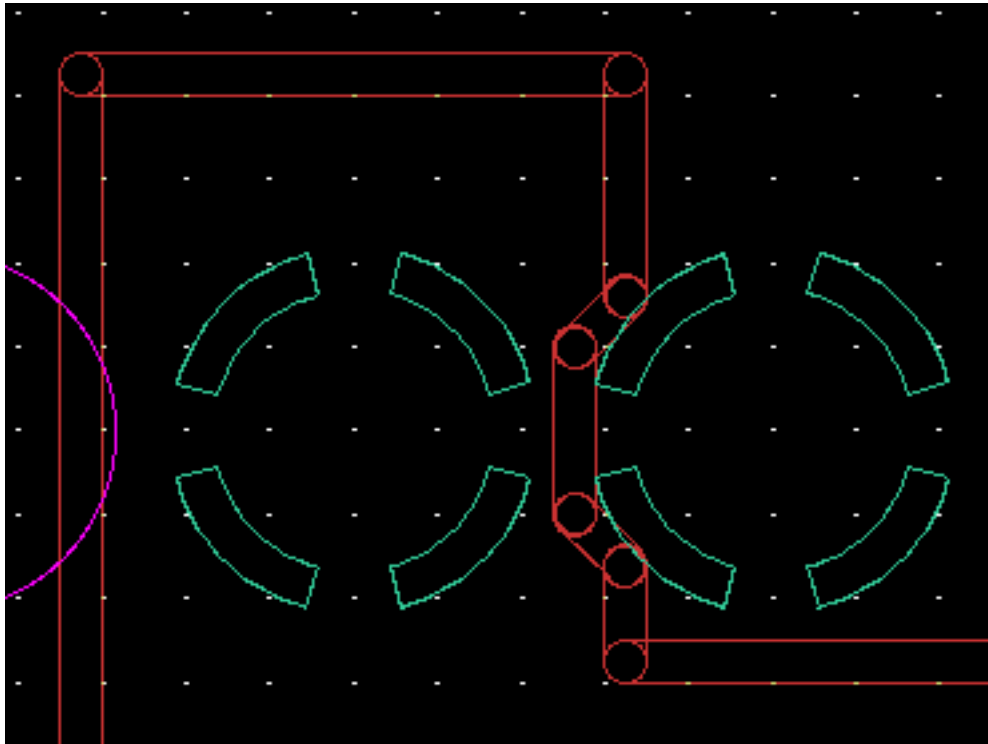
The simulation of artwork\_5 displays in a new window named E\$artwork\_5.sim, which covers the window displaying the board. This artwork is for the layer containing the GROUND and -15V power fill areas and nets.

2. Change the view area so you can see the -15V power fill area, as shown in [Figure 2-18](#). Notice how some of the thermal reliefs touch the edge of the power fill area, as shown in [Figure 2-19](#).

This is the source of the warnings about poor coverage that you saw earlier.



**Figure 2-18. Close-up of Simulated Artwork**



**Figure 2-19. Close-up Showing Thermal Relief Touching Edge of Power Fill**

3. Change the view area so you can examine the data at the bottom of the window. Also, notice that the colors associated with aperture positions match the graphics of the actual artwork data.
4. Close the E\$artwork\_5.sim window.

The edit window showing the board again becomes visible.

5. Simulate artwork\_2 so you can see the artwork for the VCC/+15V power plane.

You can see your power aperture definitions.

6. When finished with the lab, save all the design data and **Close** the FabLink session.

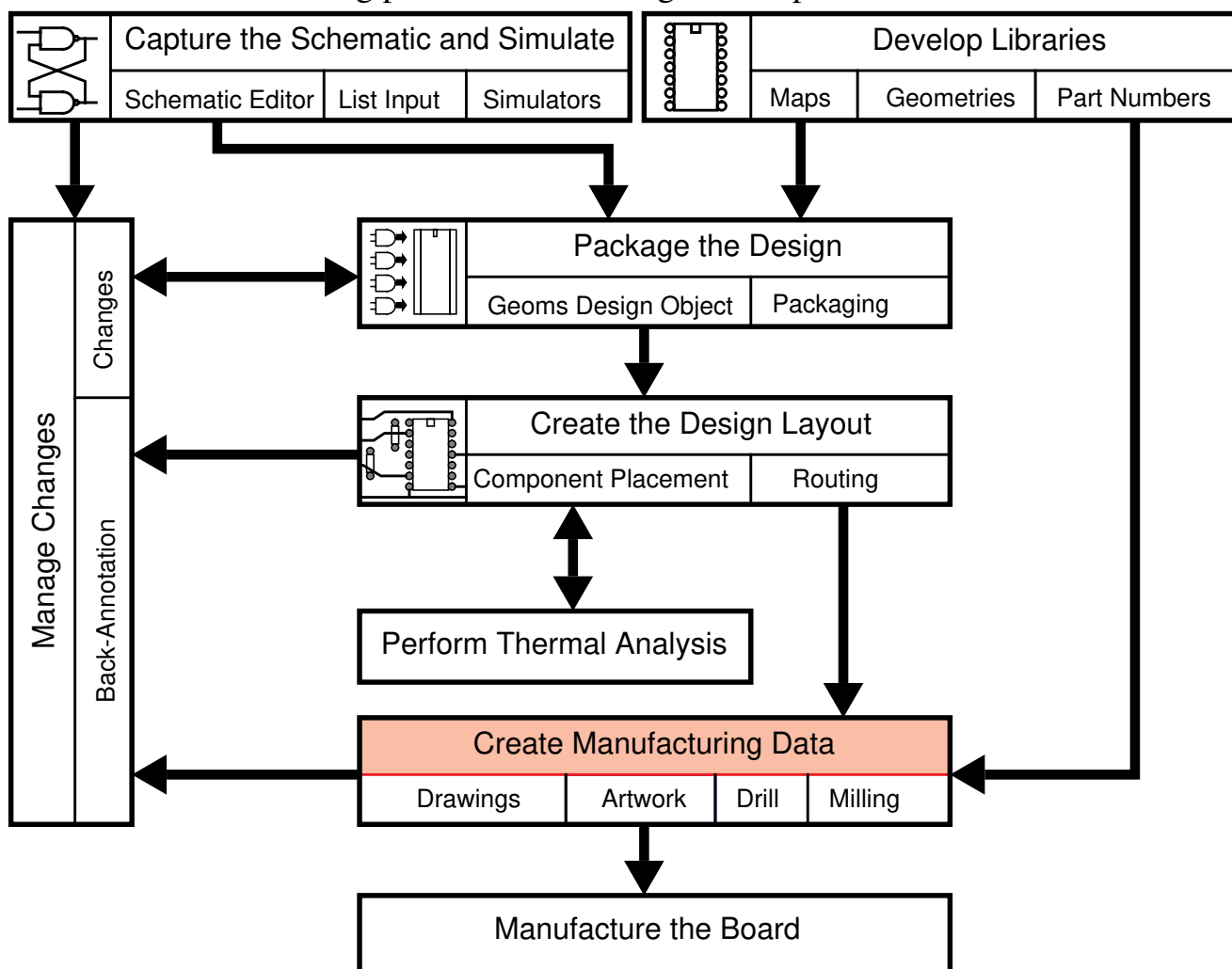
Congratulations! You have completed the "Artwork and Aperture Data" lab exercise. Continue with Lesson 3: "Creating Test Coupons and Panel Geometry".



# Lesson 3

## Creating Test Coupons and Panel Geometries

This part of the manufacturing data module explores some special manufacturing practices, including creating manufacturing panels with thieving patterns and creating test coupons.



**Figure 3-1. PCB Design Process**

## Objectives

After completing this module, you are able to describe:

- The process of creating test coupons
- The purpose of thieving patterns
- The major steps in creating thieving patterns
- The process of creating a panel

The FabLink tool assists you in the process of arranging multiple circuit board patterns (along with targets, test coupons, and other markings) onto a single artwork.

You can generate thieving patterns, in various configurations, across artwork automatically. Thieving patterns facilitate the escape of excess laminating material and are sometimes referred to as dam and venting patterns. You can also create special thieving patterns on panels. These patterns consist of the presence (dam) or absence (vent) of conductive material on the panel.

## Test Coupons

Test coupons that comply with military standards (MIL-STD-275E) are easy to create. You can choose specific sections to include, and also control the size of pads, drill holes, traces, thermal relief, and other characteristics in each coupon section. Refer to [Figure 3-2](#) and [Figure 3-3](#).

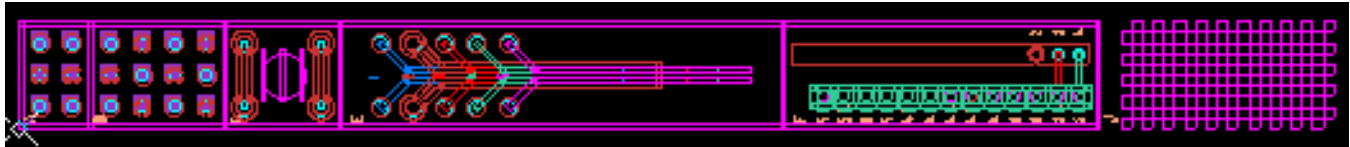


Figure 3-2. Test Coupon Example

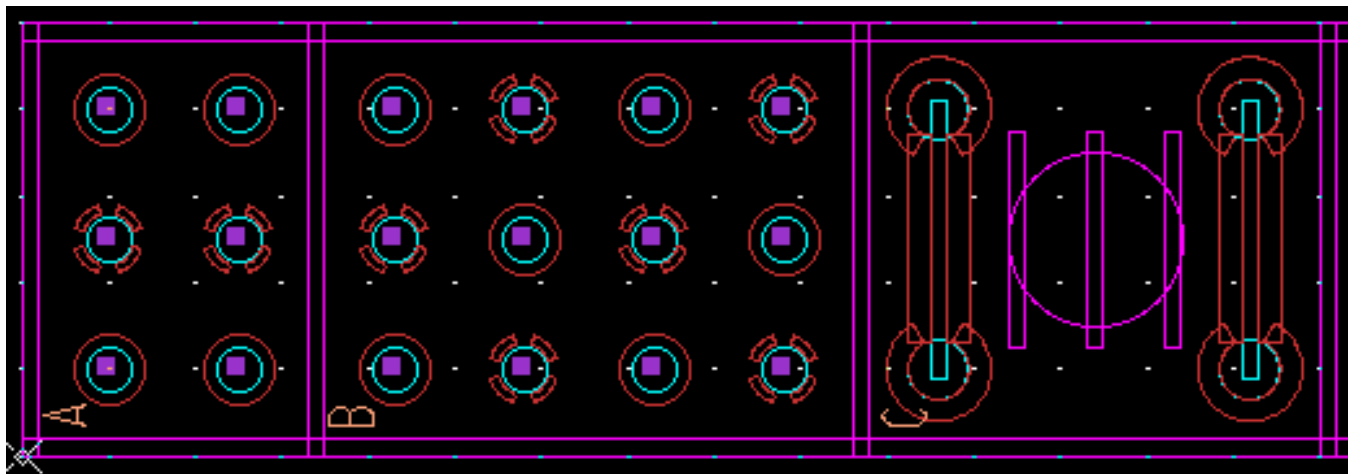


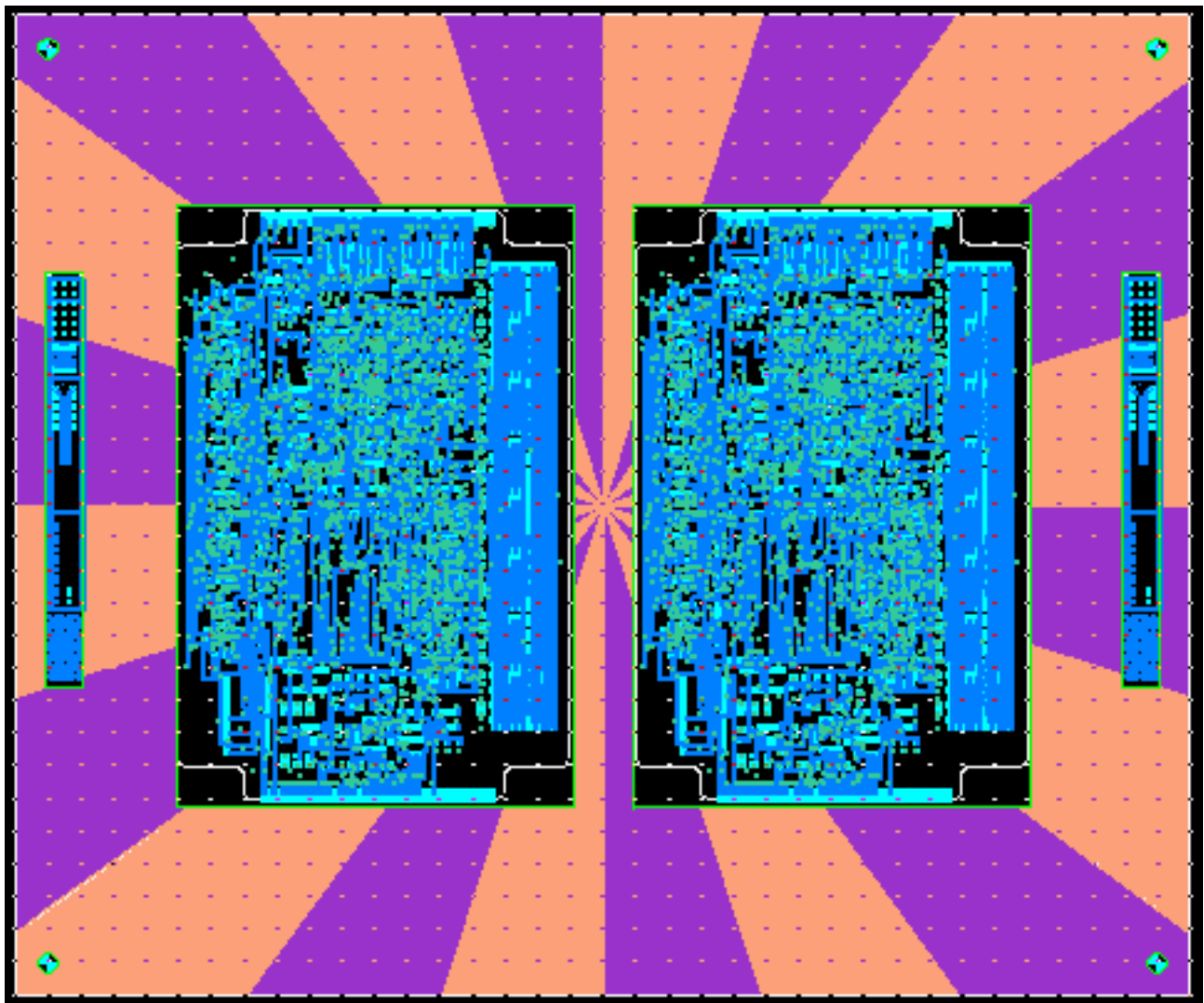
Figure 3-3. Close-up of a Coupon

To create a test coupon, choose **Geometries > Create Geometry > Test Coupons**. In the dialog box, choose the section to include in the test coupon and modify default characteristics as necessary. When you **OK** the dialog box, the master test coupon is created as a generic geometry, with each of the coupon sections that you specified. If required, you can edit any portion (padstack, text, or trace) of any section of the test coupon.

Add the test coupon to your board or panel by choosing **[Top Menu] Drawing > Extended Menu > Add Geometry**.

## The Panel Geometry

Your manufacturing facility might prefer to process several boards simultaneously by using film with multiple images of your design. A panel includes one or more board images, plus test coupons, tooling holes, and targets, as well as identification information. Additionally, a panel might include the specification for venting or thieving patterns to aid in the manufacture of the circuit board. Refer to [Figure 3-4](#).



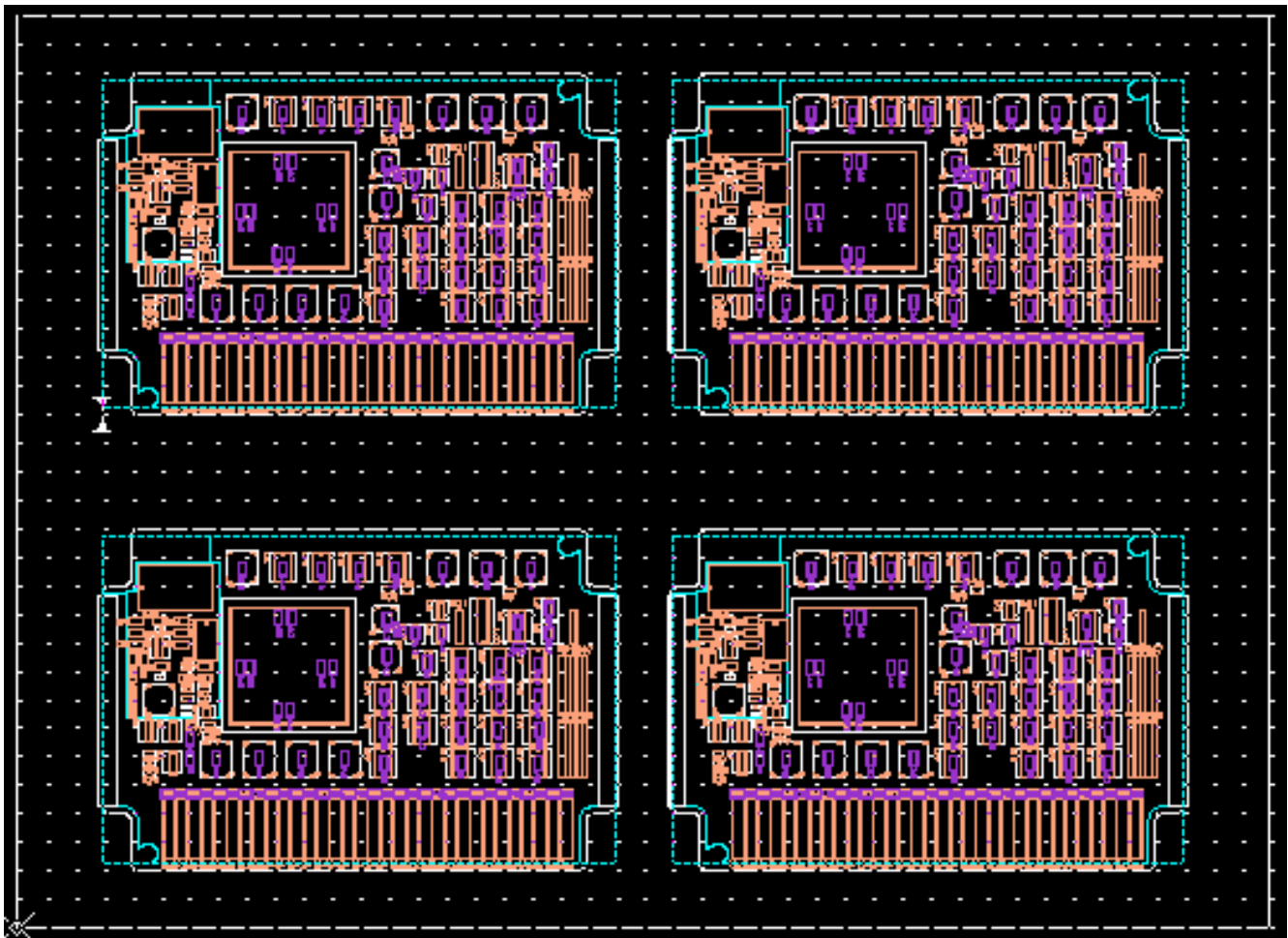
**Figure 3-4. Example of a Panel with Venting Pattern**

To create a panel geometry, choose **Geometries > Create Geometry > Manufacturing Panel**.



## Creating a Step-and-Repeat Panel

A step-and-repeat panel is one in which two or more copies of the same board layer are arranged so that one plot file can be created and the board layer is plotted several times on one piece of film. Refer to Figure 3-5.

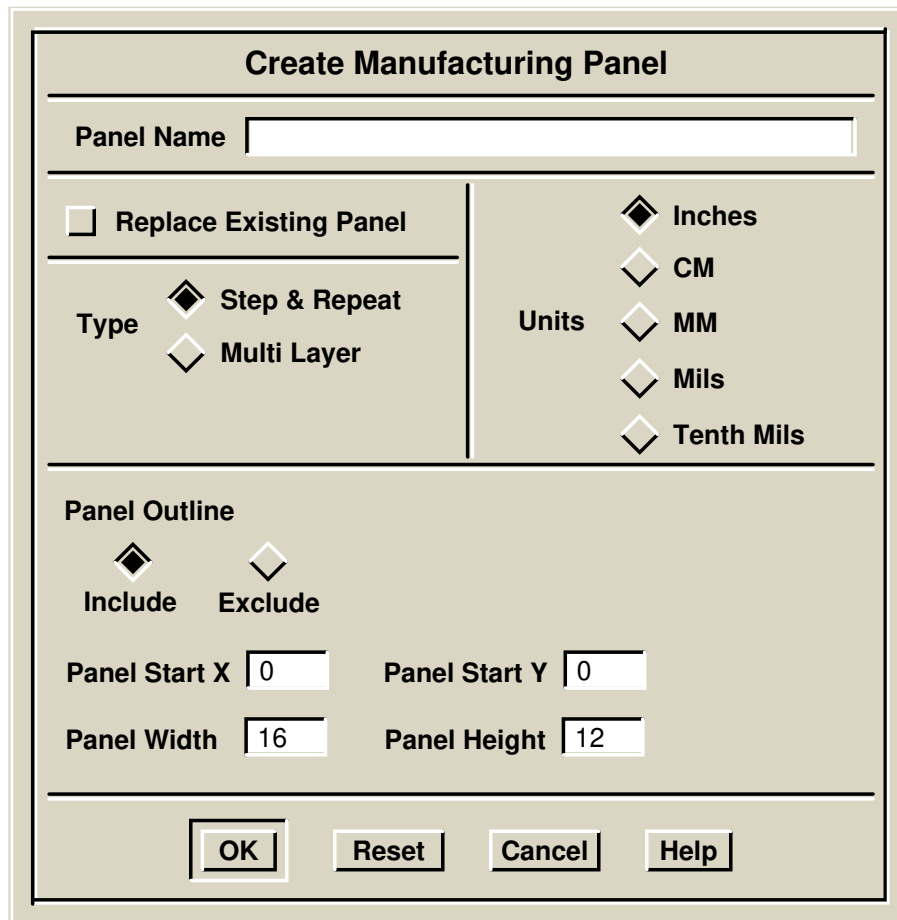


**Figure 3-5. Example of a Step-and-Repeat Panel**

Create a step-and-repeat panel as follows:

1. Draw your board geometry (if it is not already drawn) in the Edit window by choosing **Geometries > Open Geometries** and completing the dialog box.
2. Choose **Geometries > Create Geometry > Manufacturing Panel**.

A dialog box appears, as shown in [Figure 3-6](#).



The dialog box is titled "Create Manufacturing Panel". It contains the following fields and controls:

- Panel Name:** A text entry field.
- Replace Existing Panel:** A checkbox.
- Type:** Two radio buttons: "Step & Repeat" (selected) and "Multi Layer".
- Units:** Five radio buttons: "Inches" (selected), "CM", "MM", "Mils", and "Tenth Mils".
- Panel Outline:** Two radio buttons: "Include" (selected) and "Exclude".
- Panel Start X:** A text entry field with the value "0".
- Panel Start Y:** A text entry field with the value "0".
- Panel Width:** A text entry field with the value "16".
- Panel Height:** A text entry field with the value "12".
- Buttons:** "OK", "Reset", "Cancel", and "Help".

**Figure 3-6. Create Manufacturing Panel Dialog Box**

**3.** Fill in the dialog box as follows:

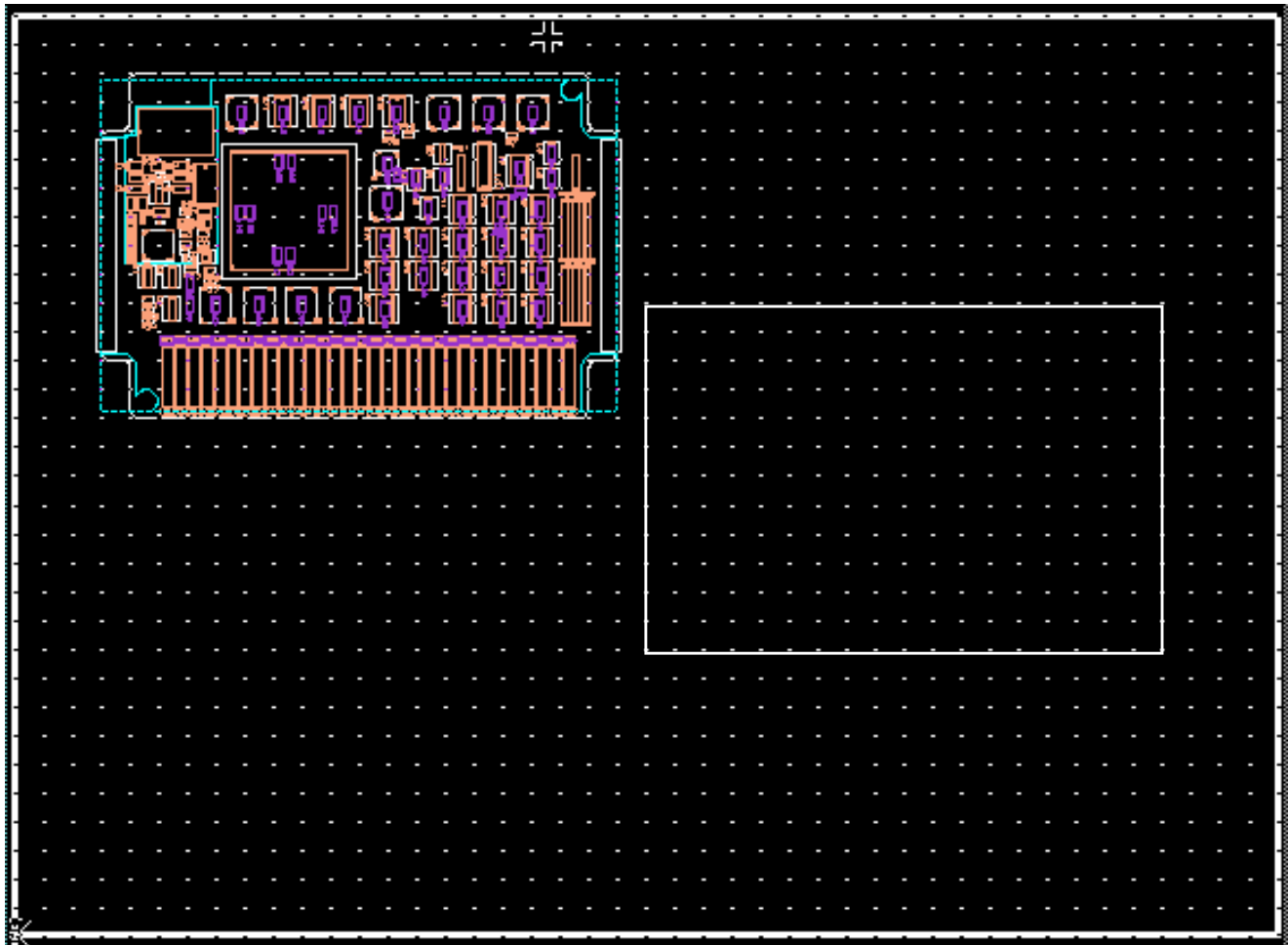
- In the Panel Name entry box, type a panel geometry name.
- Next to the Type label, make sure that Step & Repeat is selected.
- Next to the Units label, choose the type of units used in the design.
- If panel\_outline is defined as a layer in the design, the dialog box contains a Panel Outline area for specifying the panel size and also for determining whether or not you want the panel outline included. If the design has a panel\_outline layer defined, choose Include or Exclude and specify the panel size.
- Fill out the dialog box and press **OK** to display the defined panel.

4. If you chose to include the panel outline, the view is automatically zoomed out for you to see the entire board. If you chose to exclude the panel outline, and you want to see the entire board when added to the panel, you might need to choose **View > Zoom Out**.
5. Next, use **Setup > Grid** to specify a wide grid such as 0.5 (x and y values in inches) with an interval of 1.

A wide grid setting lets you to see the grid while you view the entire sheet.

6. Specify the layers you want to view.
7. Determine how many board images you want to add to the panel, and where. Then choose **Drawing > Add Board**. Fill in the prompt bar that appears, as follows:
  - In the Geometry entry box, make sure that the name of the board geometry you want to add to the drawing appears.
  - In the Reference entry box, you can optionally specify the text used to identify the added geometry on the drawing.
  - You can choose to rotate the added geometry (to 90 degrees) with the Rotation stepper button. If you want the added geometry reflected, set the mirror/nomirror stepper button to *mirror*.
  - Leave the scale entry box setting at *1*. When you are creating panels for artwork, scale values greater or less than one (1) can be applied only to geometries other than the board.
8. Choose the location control and then move the cursor to where you want the basepoint of the board on the panel and click the Select mouse button.

Refer to [Figure 3-7](#) to see what the first placement of the board, and the ghost-image that moves with the cursor, might look like as you position the second image.



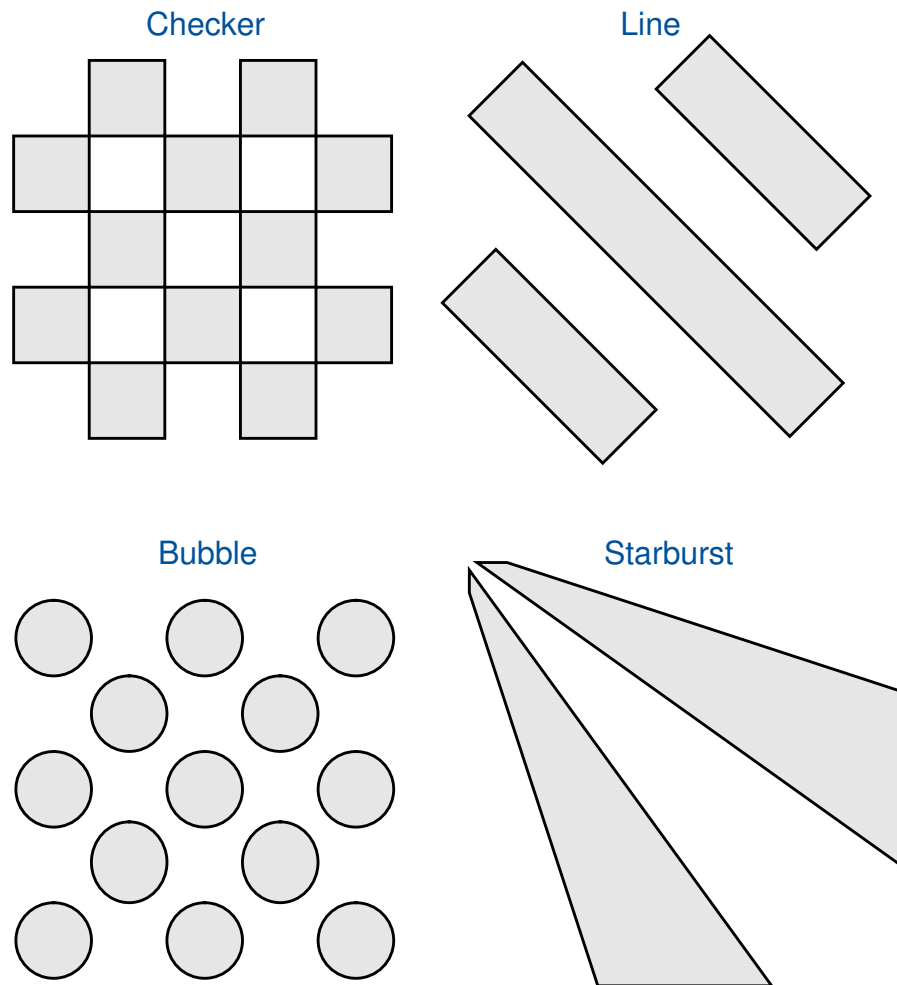
**Figure 3-7. Placing a Board on a Step-and-Repeat Panel**

The board is placed at that location.

9. Click the Menu mouse button while pressing the Shift key, or use the Execute Last Menu stroke using the Stroke mouse button. This causes the menu item **Drawing > Add Board** to repeat. Click the Select mouse button at the location for the next board. Repeat this step until you have placed as many boards as you want. Add any other information you need on the panel (for example, mounting holes, test coupons, text, and so on).
10. Choose **File > Save > Design > Geometries** to save the panel geometry.

## Thieving Patterns

The even distribution of copper during the plating process is sometimes disturbed by the presence of large areas of conductor on a panel. Thieving patterns, applied to the area surrounding the panel, offer a more uniform distribution of conductive area to aid the plating process. Refer to [Figure 3-8](#).



**Figure 3-8. Examples of Thieving Patterns**

When the individual layers of multi-layer panels are laminated together, excess laminating material must be able to escape as the layers are pressed together. The problem of material flow becomes significant for large panels. To help alleviate this problem, a pattern formed by the alternating presence and absence of conductive material

is used on each panel layer to channel the flow of laminating materials. Typically, the pattern is reversed on adjacent panel layers to ensure that pressure is applied uniformly during lamination. This process is usually called venting.

You can include a pattern for thieving and a pattern for venting in the panel description. The patterns must be cut away wherever there is a board or other object on the artwork. This is accomplished with a polygonal region called an Artwork Void. You can assign an Artwork Void to the board or panel geometry.

## Creating Thieving Patterns

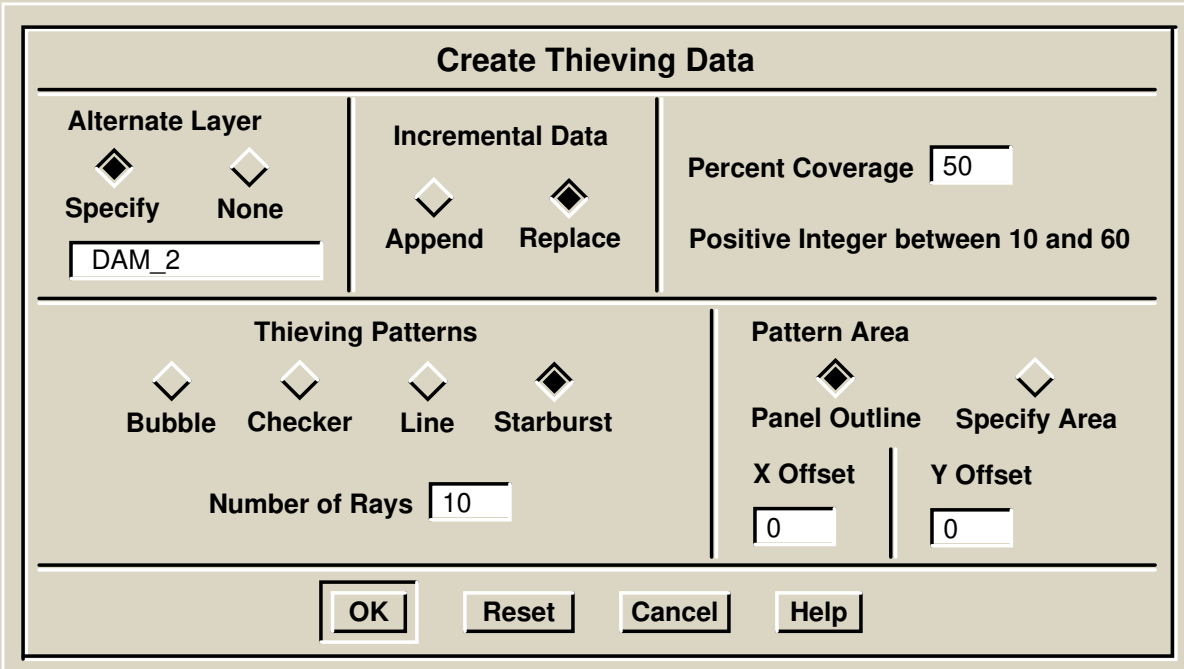
Before creating thieving patterns, you need to add an artwork void outline either to the board geometry or around the board image on the panel. The artwork void outline prevents the thieving pattern from being created over the board image, test coupons, and other images.

If you add the artwork void outline to the board geometry, the outline is automatically included with each image of the board on the panel. If you add the outline to the panel, you might have to add it around each image.

If spacing between board images is very close in the panel, you can create one artwork void outline around several images to avoid narrow bands of copper that might remain between artwork void outlines if you had made separate outlines for each image. Choose **[Top Menu] Change this Geometry > Add Artwork Void Outline**.

To create thieving data, choose **[Top Menu] Drawing > Add Thieving Pattern**. A dialog box appears, as shown in [Figure 3-9](#).

Be sure to modify your artwork order so that each artwork layer includes a layer where the thieving pattern is defined.



The dialog box is titled "Create Thieving Data". It is divided into several sections:

- Alternate Layer:** Contains two radio buttons, "Specify" and "None". The "Specify" button is selected. Below it is a text entry box containing "DAM\_2".
- Incremental Data:** Contains two radio buttons, "Append" and "Replace". The "Replace" button is selected.
- Percent Coverage:** A text entry box containing "50". Below it is the text "Positive Integer between 10 and 60".
- Thieving Patterns:** Contains four radio buttons: "Bubble", "Checker", "Line", and "Starburst". The "Starburst" button is selected. Below these is a text entry box for "Number of Rays" containing "10".
- Pattern Area:** Contains two radio buttons, "Panel Outline" and "Specify Area". The "Panel Outline" button is selected. Below these are two text entry boxes for "X Offset" and "Y Offset", both containing "0".

At the bottom of the dialog box are four buttons: "OK", "Reset", "Cancel", and "Help".

**Figure 3-9. Create Thieving Data Dialog Box**

- **Alternate layer**—Places the pattern on the Dam\_1 layer. To create an alternating pattern, select the Specify radio button and type Dam\_2 in the entry box.
- **Incremental Data-Append/Replace**—Press the Append button to add a new thieving pattern to an old thieving pattern, or press the Replace button to replace any existing thieving pattern.
- **Percent Coverage**—Specifies the percentage of the panel to cover with copper. This value is also used to determine the distance between pattern elements (dots, checkers, or lines).
- **Thieving Patterns**—Specifies the type of pattern. You can also specify the size of the pattern.
- **Pattern Area-Panel Outline/Specify Area**—Press the Panel Outline button to place the pattern within the area defined by a path on the Panel\_outline layer. If you do not want to place the pattern within the Panel\_outline path, you can specify the area for the pattern by pressing Specify Area and then entering the x,y locations of the lower-left and upper-right corners of the area.

## Creating Panel Artwork Data

Creating artwork data for a panel is similar to creating artwork data for a board.

- If you have not defined the aperture table, choose **[Top Menu] Artwork > Change Aperture Table > Fill Aperture Table**. It is a good idea to recreate your aperture table, because new aperture sizes might be required by test coupons or other elements added to the panel geometry.
- Generate artwork for the panel by choosing **[Top Menu] Artwork > Create Artwork Data**. In the dialog box, press the Step & Repeat Panel button (below the Output Type label). Fill in the rest of the form. Press **OK**.
- After the artwork data is generated, you can view it by choosing **[Top Menu] Artwork > Open Artwork Data**.



## Lab Exercise

This lab exercise shows you more fabrication data output features. To observe these output features, you create a panel that includes two copies of the board you completed in an earlier lab, plus test coupons and targets. As you assemble the panel data, you also construct thieving patterns, which become part of your artwork image.

Upon completion of this lab exercise you are able to:

- Create a test coupon automatically.
- Create a step-and-repeat panel geometry.
- Add the board geometry, the test coupon, and targets to the panel.
- Add the panel outline and artwork voids to the panel.
- Create thieving patterns on the panel.
- Verify aperture table setup and create artwork data for the panel.

Turn to Module 7—Lab 3: "Creating Test Coupons and Panel Geometries".



# Lab 3

## Creating Test Coupons and Panel Geometry

### Introduction

This lab exercise shows you more fabrication data output features. To observe these output features, you create a panel that includes two copies of the board you completed in an earlier lab plus test coupons and targets. As you assemble the panel data, you also construct thieving patterns, which become part of your artwork image.

Upon completion of this lab exercise you are able to:

- Create a test coupon automatically.
- Create a step-and-repeat panel geometry.
- Add the board geometry, the test coupon, and targets to the panel.
- Add the panel outline and artwork voids to the panel.
- Create thieving patterns on the panel.
- Verify aperture table setup and create artwork data for the panel.

## Procedure

In this lab exercise you use FabLink to create panel artwork data.

## Preparation for Lab

1. Invoke the Design Manager. Invoke FabLink from the FabLink icon in the Design Manager Tools window. In the navigator dialog box, navigate to the `your_path/training/board_new/mod7/sig_az` design and select it, then **OK** the dialog box.
2. Close the report window.

## Creating a Link to a Geometry Library

If taking this training course as part of a workshop, skip this procedure section and begin with ["Constructing a Panel" on page -17](#). In the workshop version of this training material, the libraries are set up for you by your instructor. You must create a link to the geometry library only if completing this training as a Personal Learning Program.

1. Choose **Geometries > Add Library Link**. In the dialog box, enter the following information and press **OK**.

Library Name: **mgc.trng.drawings**

Pathname to Existing Library:

`your_path/training/board_new/mod7/sig_az/pcb_parts/  
user_geom/drawings`

Add to: **User**

Directory type: **Permanent**

Now set up conditions for creating a panel.

## Constructing a Panel

To demonstrate some additional features, you need to construct a panel 18 inches wide by 15 inches high. In this lab exercise, you create the panel geometry and the test coupon. In the panel, you include two rotated images of the board, a test coupon to the left and right of the board images, targets in each corner, and a starburst thieving pattern. You completed the board geometry earlier. The target is provided in a user library.

1. Set the grid to **0.5** inches with a display interval of **1**.
2. Set the line width to **.010** inches.
3. Set the snap direction to **Any Angle**.
4. View only the layers: **Board\_outline, Dam, Dam\_1, Dam\_2, Pad\_1, Panel\_outline, Signal, Signal\_1** and **Via**.
5. List the geometry libraries, view the contents of the user library **mgc.trng.drawings**, and then read the geometry **target\_a**.

A new edit window displays containing the target. A report window might also be created.

6. Close the report window and the new target edit window.

After you supply format specifications, FabLink automatically creates test coupons for you. In the next step, you supply specifications for the test coupons.

7. Choose **Geometries > Create Geometry > Test Coupons...** Enter the information from [Table 3-1](#) into the Create Test Coupons dialog box and press **OK**.

**Table 3-1. Create Test Coupons Dialog Box Information**

Test Coupon Geometry Name: <b>test_cpn</b>	
<b>COUPON A: Include</b> Order: <b>1</b> Pad Size: <b>0.05</b> Clearance Size: <b>0.08</b> Hole Size: <b>0.028</b> Power Pad Option: <b>Draw</b> Tie Width: <b>0.02</b> Air Gap Width: <b>0.01</b>	<b>COUPON B: Include</b> Order: <b>2</b> Pad Size: <b>0.05</b> Clearance Size: <b>0.08</b> Hole Size: <b>0.028</b> Power Pad Option: <b>Draw</b> Tie Width: <b>0.02</b> Air Gap Width: <b>0.01</b>
<b>COUPON C: Include</b> Order: <b>3</b> Pad Size: <b>0.07</b> Hole Size: <b>0.028</b> Trace Clearance: <b>0.025</b>	<b>COUPON J: Include</b> Order: <b>6</b>
<b>COUPON E: Include</b> Order: <b>4</b> Pad Size: <b>0.07</b> Clearance Size: <b>0.08</b> Hole Size: <b>0.025</b> Trace Clearance: <b>0.025</b>	<b>COUPON F: Include</b> Order: <b>5</b> Pad Size: <b>0.05</b> Hole Size: <b>0.028</b> Trace Size: <b>0.02</b> Test Clearance Size Range: <b>0.032 0.056</b>

It takes FabLink a few moments to create the several pieces of required geometry, each with its own edit window. When it finishes, FabLink automatically constructs a test coupon based on the specifications, and places the test coupon in an edit window.

8. In the test\_cpn edit window, use the View All stroke to view the test coupon.

When you first press the Stroke mouse button, the window might flicker a little as the edit window is activated.

9. Close the test coupon edit window when you are done viewing it.

You are now ready to begin panel construction.

10. Choose **Geometries > Create Geometry > Manufacturing Panel...**  
Enter the following in the dialog box, then press **OK**.

Panel Name:	<b>mfg_panel</b>		
Type:	<b>Step &amp; Repeat</b>		
Units:	<b>Inches</b>		
Panel Outline:	<b>Include</b>		
Panel Start X:	<b>9</b>	Panel Start Y:	<b>7.5</b>
Panel Width:	<b>18</b>	Panel Height:	<b>15</b>

A new edit window, named PA\$mfg\_panel, displays containing the panel.

11. Activate and View All of the PA\$mfg\_panel edit window to see the entire panel outline.

In the next steps, you add four targets to the panel. Later, you add the boards and test coupons. The process of adding geometry is the same in FabLink as it is in LIBRARIAN.

12. Add the first target to the upper-right corner of the board by choosing **[Top Menu] Shapes > Extended Menu > Add Geometry**. In the prompt bar, enter the following information. Tab to the location prompt.

Geometry: **target\_a**  
Reference: **tg1**

You entered a reference name to make it easier to locate the item if you decide to create an ASCII *geoms* design object later. The reference is tg1.

If you move the cursor in the edit window, you see a ghost image of the target.

13. Place the target 0.5 inch inside the upper-right corner of the panel (at absolute coordinate X=26.5, Y=22). Refer to [Figure 3-10](#) for an illustration of the completed panel.

The Add Geometry prompt bar repeats so you can add the next target.

14. Enter the following values into the prompt bar, Tab to the location prompt, and add the second target in the lower-right corner of the panel at absolute coordinate X=26.5, Y=8.

Geometry: **target\_a**  
Reference: **tg2**

The prompt bar repeats.

15. Enter the following values into the prompt bar, Tab to the location prompt, and add the third target in the upper-left corner of the panel at absolute coordinate X=9.5, Y=22.

Geometry: **target\_a**  
Reference: **tg3**

The prompt bar repeats.



16. Enter the following values into the prompt bar, Tab to the location prompt, and add the fourth target in the lower-left corner of the panel at absolute coordinate X=9.5, Y=8.

Geometry: **target\_a**

Reference: **tg4**

17. Do not cancel the Add Geometry prompt bar. While it displays, change the grid to 0.125 with a display interval of 4.

The grid is still displayed in 0.5 inch intervals. You next add the test coupon geometries. The Add Geometry prompt bar is still displayed.

18. Enter the following values in the Add Geometry prompt bar. Tab to the location prompt. Add the first test coupon 0.5 inches inside the left edge of the panel, centered between the top and bottom edges of the board, near or at absolute coordinate X=9.5, Y=18.5.

Geometry: **test\_cpn**

Reference: **test1**

Rotation: **d270**

The prompt bar repeats.

19. Enter the following in the prompt bar, Tab to the location prompt, and place the second test coupon at the right edge of the board near or at absolute coordinate X=26, Y=18.5.

Geometry: **test\_cpn**

Reference: **test2**

Rotation: **d270**

The prompt bar repeats again. Next, you add the board geometry to the panel. You place one image on the left half of the panel and a second image on the right half of the panel.

20. Enter the following in the prompt bar, Tab to the location prompt, and place the first board geometry near or at absolute coordinate X=17.5, Y=10.5.

Geometry: **signal\_analyzer**  
Reference: *image\_1*  
Rotation: **d90**

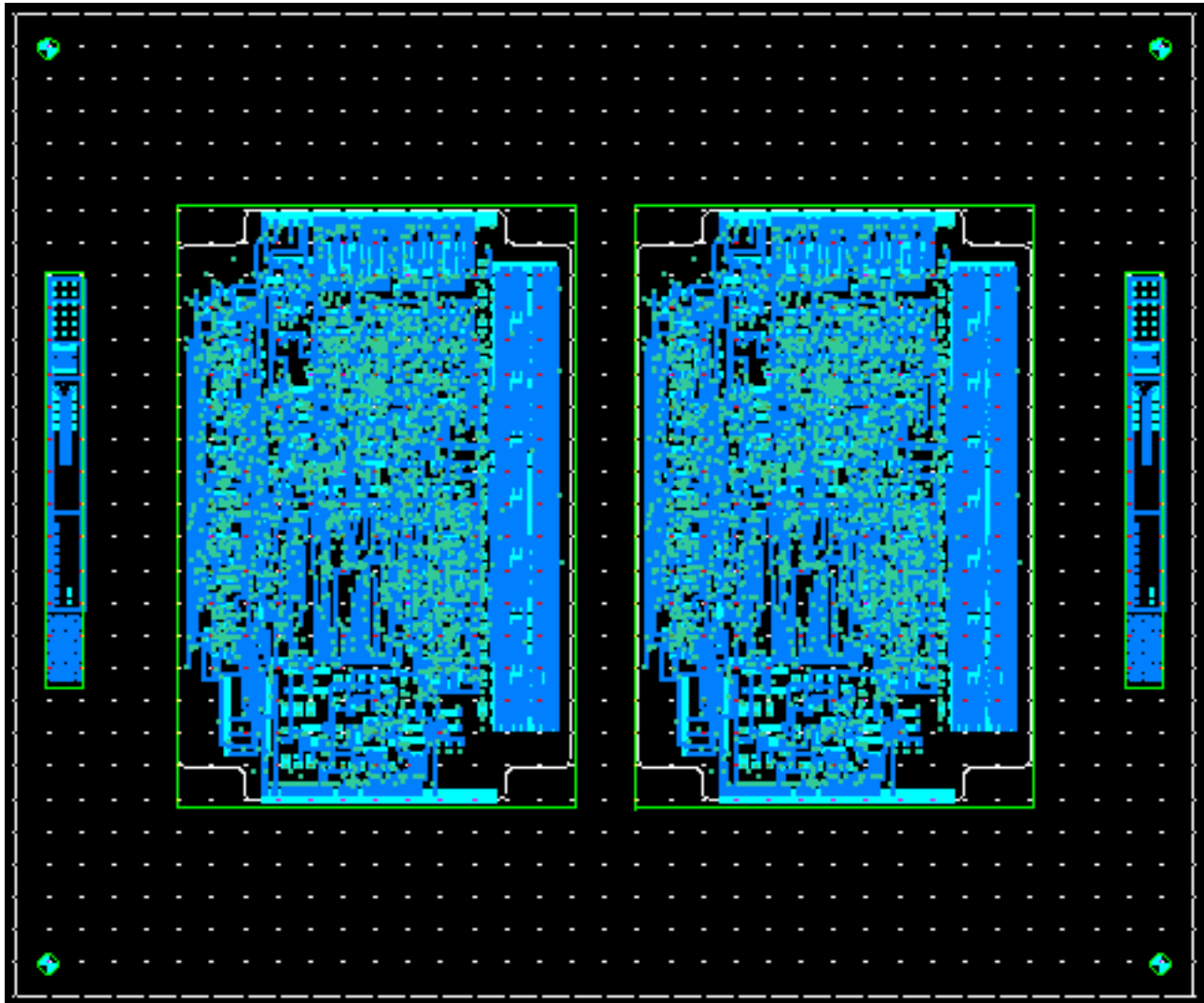
If you completed Module 4: "Creating PCB Design Geometries" of this training series, you named the board when you created the board geometry. The prompt bar repeats so you can add the other board image to the right side of the panel.

21. Enter the following in the prompt bar, Tab to the location prompt, and place the board geometry near or at absolute coordinate X=24.5, Y=10.5.

Geometry: **signal\_analyzer**  
Reference: *image\_2*  
Rotation: **d90**

The panel now looks similar to [Figure 3-10](#). (The figure shows the Artwork Void Outlines, which you have not added yet.) There is still more to add to the panel before all the manufacturing data is complete.

You need to add an Artwork Void Outline around each added feature. The Artwork Void Outlines allow FabLink to cut holes in the thieving pattern for the board images, targets, and test coupons.



**Figure 3-10. Completed Outline**

22. Cancel the Add Geometry prompt bar.
23. Use the View Area stroke to view a very small area around one of the targets, so that the target is very large in the edit window.
24. Set the grid to .025 inches, with a display interval of 4.

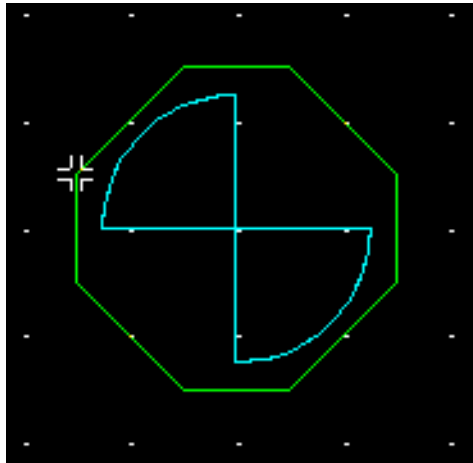
Next, you add the Artwork Void Outline around the target as an octagon-shaped polygon.

**25. Choose [Top Menu] Change This Geometry >**

**Add Artwork Void Outline.** In the prompt bar, leave the Layer prompt at the default value. Verify that the location prompt is highlighted. Click the Select mouse button to specify the location of the corners of the Artwork Void Outline shown in [Figure 3-11](#). After you specify all the points needed for the octagon, **OK** the prompt bar.



*Use the BACKSPACE key to remove or back up to the previous location if you specify an incorrect location.*



**Figure 3-11. Target With Artwork Void Outline**

The Add Artwork Void Outline prompt bar repeats.

**26. Add an Artwork Void Outline to each of the other three targets, as you did to the first. View all of the panel, and then use View Area to change the view to each of the targets.**

Alternately, you can cancel the prompt bar, select the first artwork void outline you created, and copy it to the other targets.

27. Do not cancel the Add Artwork Void Outline prompt bar when you finish creating the artwork void outlines around the targets. If you canceled it because you chose to copy the outlines, choose **[Top Menu] Change This Geometry > Add Artwork Void Outline** again.

Next, you create artwork void outlines around the test coupons.

28. Create an artwork void outline around each test coupon in the shape of a rectangle, with a clearance of .050 inches between each coupon and its outline. Do not cancel the Add Artwork Void Outline prompt bar when you finish creating the artwork void outlines around the test coupons.

Next, you create artwork void outlines around the edge of both board images.

29. Reset the grid to .125 inches with an interval of 4.
30. Add the Artwork Void Outline around the edge of both board images in the panel. Use a clearance of approximately .125 inches between each board and its outline.

You can add the Artwork Void Outline for the board image either to the board geometry when you create the board, or directly in the panel as you did in this lab. If spacing between board images is very close in the panel, you can create one artwork void outline around several images to avoid narrow bands of copper that might remain between artwork void outlines if you had made separate outlines for each image.

The method you use (either placing the outline in the board geometry or on the panel) depends on the spacing between the images in the panel. You want to avoid small slivers of copper left on the panel, which could cause problems during manufacturing.

31. Cancel the Add Artwork Void Outline prompt bar.

**32.** View all of the panel.

Your panel geometry is ready for a thieving pattern. FabLink creates thieving patterns automatically based on your specifications.

**33.** Choose [**Top Menu**] **Drawing > Add Thieving Pattern > Add Pattern...** Enter the following values in the dialog box, then press **OK**.

Alternate layer:           **Specify DAM\_2**

Incremental Data:       **Replace**

Percent Coverage:       **50**

Thieving Patterns:       **Starburst**

Number of Rays:         **10**

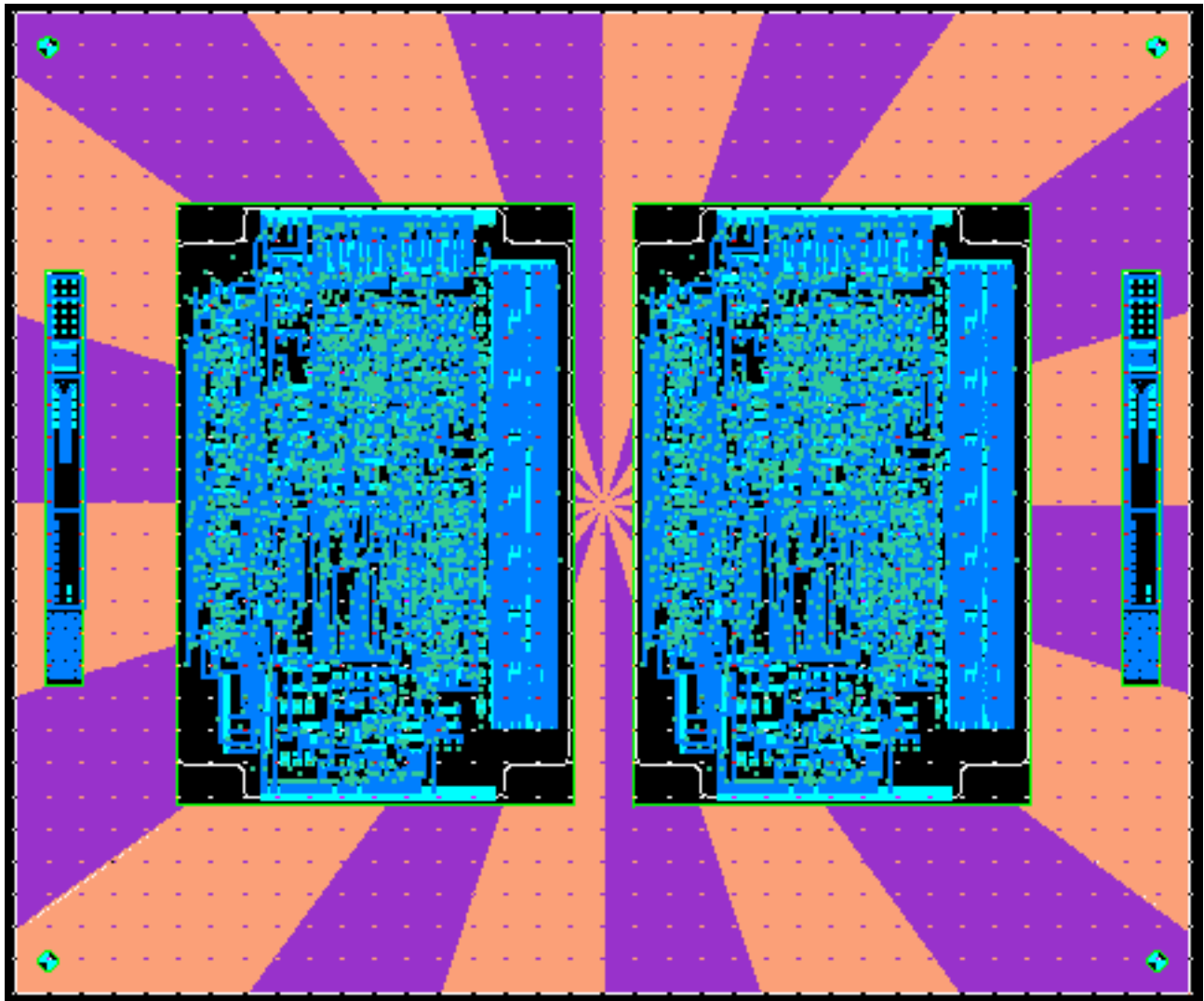
Pattern Area:            **Panel Outline**

X Offset: **0**   Y Offset: **0**

You can see the lines of the starburst pattern radiating from the center of the panel. To see the alternating layer pattern better, change your polygon view style to filled (using **View > Change View Style**).

34. Change the view style to filled (look in the View menu).

Your completed panel with thieving patterns looks similar to Figure 3-12.



**Figure 3-12. Panel With Thieving Patterns**

## Creating a Board Layer Block

FabLink creates a Board Layer Block for you automatically. A Board Layer Block is a geometry that causes physical layer numbers to appear on the artwork sheets and on the completed board. In the following discussion you add the block to your panel. Before creating the Board Layer Block, you need to set up text and visible layers.

1. Choose **Setup > Text...**, enter the following values in the dialog box, and press **OK**.

Height:	<b>.06</b>
Font:	<b>Microfilm</b>
Justification:	<b>Bottom Left</b>

All other settings work without change. The Setup Text dialog box controls the specification of text in the Board Layer Block.

2. Verify that all power and signal layers are visible.
3. Choose **[Top Menu] Shapes > Extended Menu > Add Board Layer Block:**. In the prompt bar, enter **.15** for width and **.15** for height. Tab to the location prompt. When you move the cursor into the edit window, notice the ghost image of the board layer block. Move the image into a clear area of one of the boards and click the Select mouse button.

FabLink builds the Board Layer Block and places it in the board.

4. Create another board layer block, and place it in a clear area within the other board.

Place the cursor in the edit window, hold down the Shift key, and click the Menu mouse key to have the prompt bar repeat. The prompt bar retains the setting specified in the previous step.



## Creating Artwork Data

In this procedure, you generate artwork for the completed design by creating artwork several times, changing options, and observing the results.

Up to this point in the lab exercise, your setup has been the creation of a panel. Now, you prepare to create artwork data for the panel. First, you modify the artwork order geometry. You then fill the aperture table, verify the settings in the aperture table, and create panel artwork data.

First, you open and modify the Artwork Order to handle the thieving patterns.

1. Choose **Geometries > Open Geometries**. In the dialog box, select **artwork\_order** and press **OK**.

The ST\$artwork\_order edit window is opened. This geometry contains no graphic data; it is a set of specifications that control the generation of Gerber artwork data. Next, you modify these specifications.

2. Choose **[Top Menu] Change This Geometry...** In the dialog box, select logical layer 1 and choose **Change**.

The Change Artwork Layer dialog box displays.

3. In the **Include Layer Names** field, add **DAM\_1** to the list of names by first backspacing one character, re-entering the character, and then entering **DAM\_1**. **OK** the dialog box. Read the following paragraph for more help.

If you try to simply type additional text into a field that already contains text, the pre-existing text is sometimes removed. To retain existing text, backspace over the last character, re-enter the character, and then enter the new text, or place the cursor after the last character in the window and click the Select mouse button.

After you **OK** the dialog box, you return to the Change Geometry Artwork Order dialog box.

4. In the Change Geometry Artwork Order dialog box, select logical layer 2 and change it to add **DAM\_2** to the list of included layer names. Choose **Add Board Edge**. Set the clearance to .05 inches. **OK** the dialog box.
5. Change layer 3 to add **DAM\_1** to the list of included layer names.
6. Change layer 4 to add **DAM\_2** to the list of included layer names.
7. Change layer 5 to add **DAM\_1**. Choose **Add Board Edge**. Set the clearance to .05 inches.
8. Change layer 6 to add **DAM\_2** to the list of included layer names.

In the dialog box, you can see that DAM\_1 and DAM\_2 alternates with each logical layer.

9. **Close** the Change Geometry Artwork Order dialog box, and also close the ST\$artwork\_order edit window.
10. If the mfg\_panel edit window is not visible, pop the edit windows to make the mfg\_panel edit window visible.
11. Choose **[Top Menu] Artwork > Change Aperture Table > Fill Aperture Table**. Leave the default settings in the dialog box and press **OK**.

A report window displays. You might see some warnings about undefined pin padstacks; you can ignore these.

12. Close the report window.

Next, you create a panel image artwork data for layer one (Signal\_1).

13. Choose [**Top Menu**] **Artwork > Create Artwork Data...** Enter the following values in the dialog box and press **OK**.

File Format: **Gerber**  
Character Set: **ASCII**  
Output Type: **Step & Repeat Panel**  
Artwork Number: **Specify** Number: **1**

**Tear Drops not Allowed**  
Thermal Reliefs in Area Fill: **On All Pads**  
**Output All Pins**  
**Output All Vias**  
**Output Unplated Holes**

Resize Artwork: **No**  
Rescale Artwork: **No**

A report window displays.

14. Close the report window.

Next you view the artwork you created.

15. Choose [**Top Menu**] **Artwork > Open Artwork Data...** Select **artwork\_1** in the dialog box and press **OK**.

A new window, named AR\$artwork\_1, displays showing the artwork data for the top of the board.

16. When you finish viewing the AR\$artwork\_1 edit window, close it.

## Simulating Artwork Data

Simulating artwork allows FabLink to imitate the photoplotting of artwork data. The output of the simulation displays in an edit window. The portions of the artwork created with each aperture display on different layers in different colors.

1. Choose **[Top Menu] Artwork > Simulate Artwork Data...** In the dialog box, select **artwork\_1** and press **OK**.

A new edit window, named E\$artwork\_1.sim, displays.

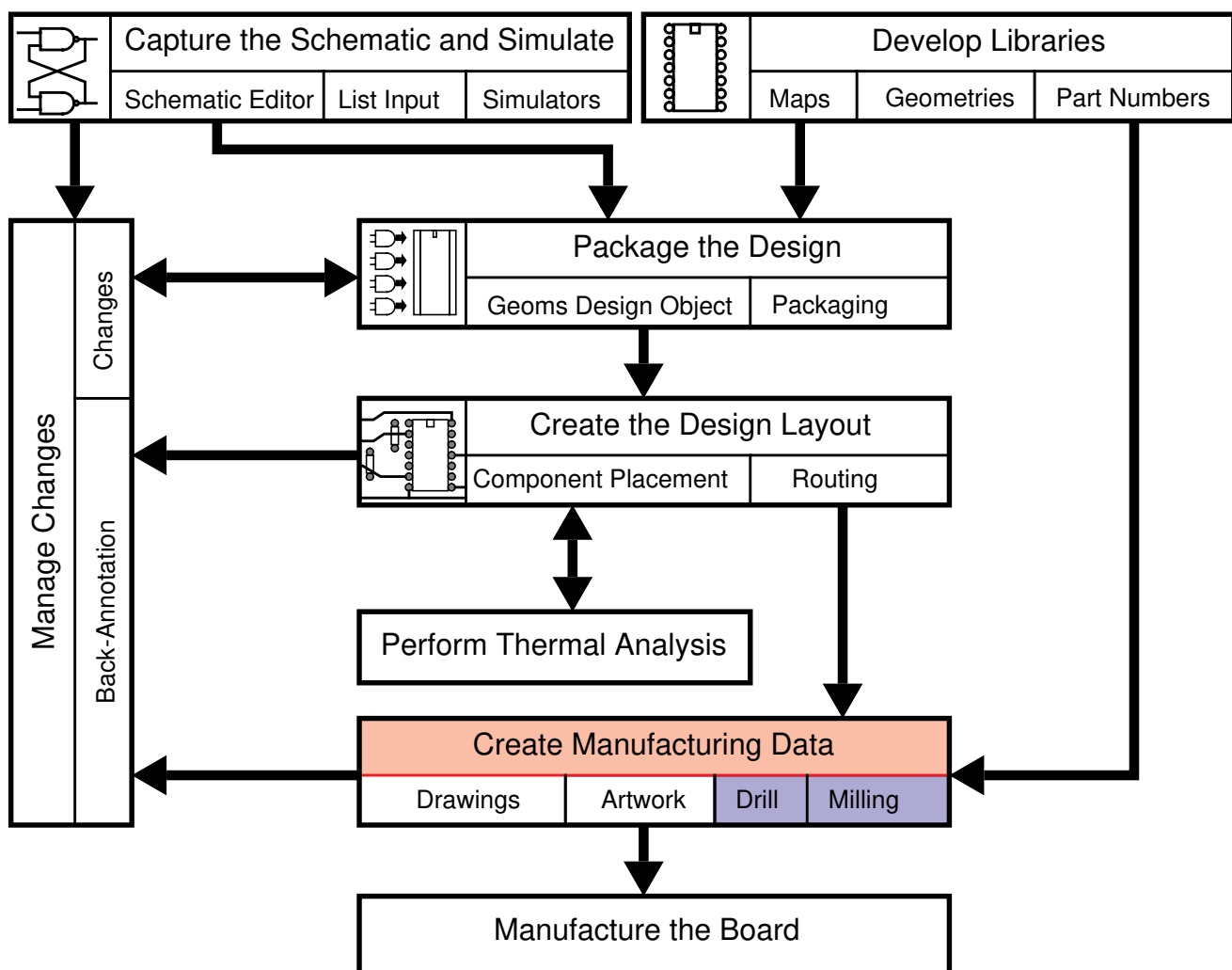
2. View the area around the text below the panel. The text includes statistics about the board.
3. Choose **File > Save > Design All** to save the design data. Then **Close** the FabLink session.

Congratulations! You have completed the "Creating Test Coupons and Panel Geometry" lab exercise. Continue with Lesson 4: "Creating Drilling and Milling Data".

## Lesson 4

# Creating Drilling and Milling Data

In this part of the manufacturing data module, you learn to create and edit drill tables, and to create drill data and milling paths.



**Figure 4-1. PCB Design Process**

## Objectives

After completing this module, you are able to describe:

- The purpose a drill table
- The steps used to create or change a drill table
- The process of creating drill data
- The steps necessary to create a milling path

The creating of numerical control (NC) data for drilling and milling is a process similar to the process of creating artwork data. In this submodule, you examine the basic process of performing the setup and creation of NC data for drilling and milling of printed circuit boards.

## Drill Table Functions

A drill table correlates pin or via padstack hole sizes, as well as tooling and mounting hole sizes, with the NC drill machine's tool position numbers. For every drill hole required in the design there must be a matching drill bit.

Before you create NC drill data, you must define a drill table.

## Defining and Saving the Drill Table

The drill table is an ASCII file created in FabLink that associates drill holes in your design with drill bits in the drill magazine. The process of defining the drill table, also referred to as setting up the drill magazine, includes the following:

- To let the system automatically fill the drill table with all the drill sizes required for your design, choose the **[Top Menu] Drill > Change Drill Table > Fill Drill Table** popup menu item.
- If you are creating a new drill table, then choose the **[Top Menu] Drill > Change Drill Table** popup menu item. If you already defined or restored a drill table in the current session, the current drill table definitions appear in the dialog box.
- To save the drill table, choose one of the following menu items:
  - **File > Save > Design All**—saves the entire design, including the drill table, as design objects.
  - **File > Save > Design Specify...**—displays a dialog box from which you can choose to save the drill table as a design object. The drill table is saved in the default location in your design (*design/pcb/drill\_table*).
  - **File > Save > to File...**—displays a dialog box from which you can choose to save the drill table as an ASCII file at any location you specify.
  - **File > Save > to Design Object...**—displays a dialog box from which you can choose to save the drill table as a design object in your design/*pcb* container with a name you specify.

## Restoring a Previously Saved Drill Table

You can restore a previously defined and saved drill table by choosing one of the following:

- **File > Restore > from Design...**—displays a dialog box listing all objects belonging to the design that can be restored. You can choose to restore the drill table.
- **File > Restore > from File...**—displays a dialog box from which you choose to restore the drill table, and you specify the pathname to the ASCII drill table.
- **File > Restore > from Design Object...**—displays a dialog box from which you choose to restore the drill table. Use this menu item if you saved the drill table as a design object with a name other than the default.

## Producing Drill Data

After you define your drill table and drill format, you are ready to create drill data, as follows:

1. Choose **Check > Drill Table** to check that your drill table has all necessary positions defined.
2. Choose **Report > Drill Format** to see a listing of the drill machine data definitions.
3. Choose the [Top Menu] **Drill > Create Drill Data** popup menu item in the board geometry edit window. Fill in the dialog box (shown in [Figure 4-2](#)) according to the specific drill output you want, as described in the following steps.
4. Below the Drill File Format label, choose Excellon or Trudrill\_c500, or choose the Specify option and enter a format in the Format entry box.
5. Below the Generate for label, specify either Board or, if you have already defined a panel, Step & Repeat Panel.



**Create Drill Data**

---

**Drill File Format**

☒ Excellon  
 ☐ Trudrill\_c500  
 ☐ Specify

**Generate for**

☒ Board  
 ☐ Step & Repeat Panel

---

**Drill Character Set**

☒ ASCII  
☐ EBCDIC  
☐ EIA

**Drill Hole Types**

☐ Unplated Thru-Holes  
 ☒ Plated Holes

**Defaulted Layer Pairs**

From Layer	To Layer	Generate	
Trace_Layer_1	VCC	Yes	0
Trace_Layer_2	Trace_Layer_3	Yes	1
ground	Trace_Layer_4	Yes	2
Trace_Layer_1	POS15V	Yes	3
NEG15V	Trace_Layer_4	Yes	4
Trace_Layer_1	Trace_Layer_2	Yes	5
Trace_Layer_3	Trace_Layer_4	Yes	6
Trace_Layer_1	NEG15V	Yes	7
POS15V	Trace_Layer_4	Yes	8
Trace_Layer_1	ground	Yes	9
VCC	Trace_Layer_4	Yes	10
Trace_Layer_1	Trace_Layer_4	Yes	11

Generate All Layer Pairs
Generate Layer Pair

---

OK
Reset
Cancel
Help

**Figure 4-2. Creating a Drill Table**

6. Below the Drill Character Set label, specify ASCII, EBCDIC, or EIA. The only drill character set that you can later view is ASCII. If you need to produce EBCDIC or EIA, generate the data first in ASCII. After it is viewed and found correct, generate the data again in either of the other character sets.
7. Below the Drill Hole Types label, choose Unplated Thru-Holes or Plated Holes.
8. After reviewing your choices, press **OK** to create your drill data.

## Drill Data File Naming

If you specify Plated Holes, the drill output for all through-vias, through-pins, and plated tooling holes is saved in a file named *drill* in your design/*pcb/mfg* directory. If you specify Unplated Thru-Holes, then data for unplated drill holes is saved in a file named *drill\_unplt* located in your design/*pcb/mfg* directory.

If your design contains blind pins and/or blind or buried vias, and you select Plated Holes, then the Generate All Layer Pairs and Generate Layer Pair buttons are available.

Pressing the Generate All Layer Pairs button generates drill files for pins and through-vias *as well as* buried, blind, and segmented vias.

The naming convention for drill files produced when you choose Generate All Layer Pairs follows this pattern (this example is for a six-layer board):

- Vias and blind pins from layers 1 to 2 are output to a file named *drill\_1\_2*.
- Vias from layers 2 to 3 are output to a file named *drill\_2\_3*.
- Vias from layers 2 to 4 are output to *drill\_2\_4*.
- Vias and blind pins from layers 5 to 6 are output to *drill\_5\_6*.
- Through-holes from layers 1 to 6 are output to a file named *drill\_1\_6*.

The drill data files are placed in your design/*pcb/mfg* directory.

To create drill files for only specified layer pairs, select the layer pair from the list box before pressing the Generate Layer Pair button. The listing for this layer pair (shown under the Generate column) toggles from No to Yes (or vice versa). You can set the state of all layer pairs by repeating this process.

## Assigning Drill Symbols to Drill Sizes

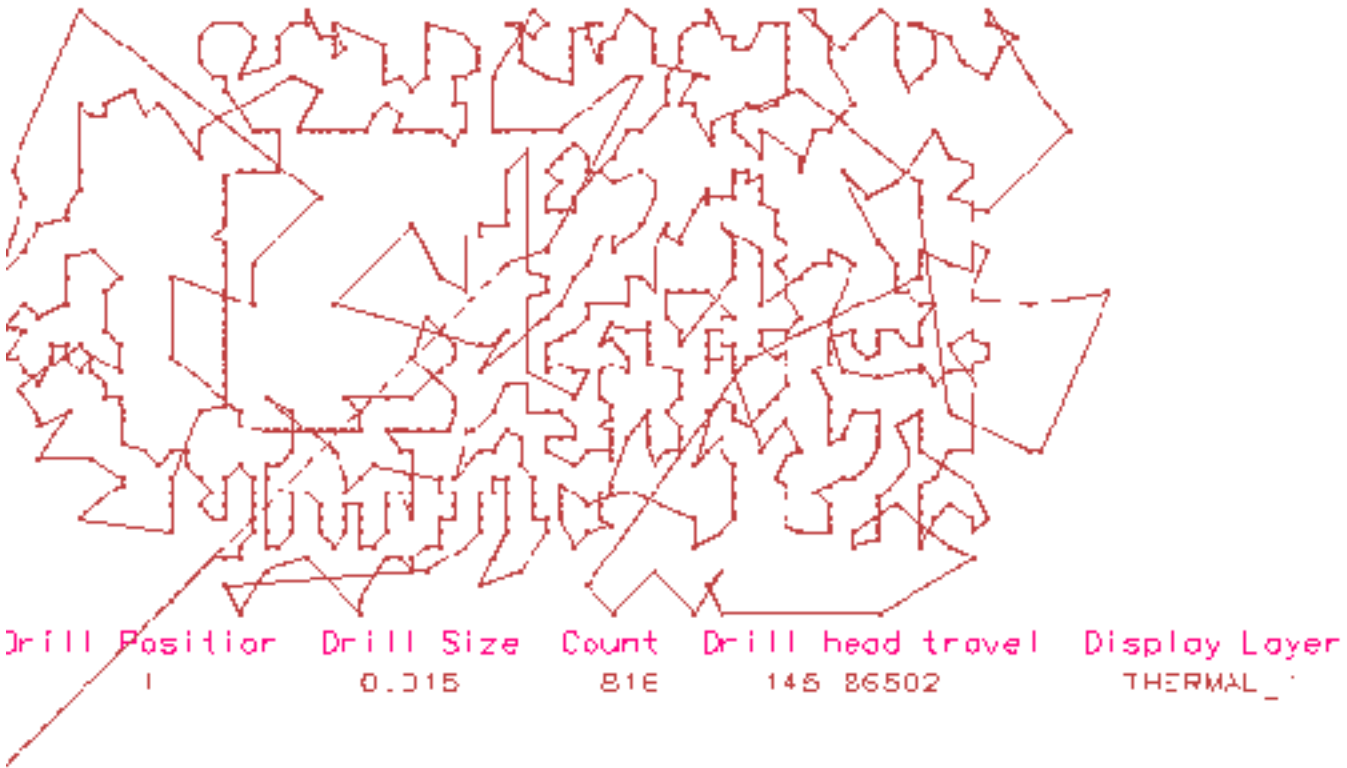
You have the option of assigning drill symbols to specific drill sizes when you set up the drill table. This allows you to standardize on a consistent drill symbol convention, making fabrication drawings easier to read. The last column of [Table 4-1](#) lists symbols, which are the names of LIBRARIAN geometries, assigned to each drill.

**Table 4-1. Example Drill Table**

<b>Drill Position</b>	<b>Drill Size</b>	<b>Upper Bound</b>	<b>Lower Bound</b>	<b>Feed Rate</b>	<b>Speed Rate</b>	<b>Plated</b>	<b>Symbol</b>
1	0.028000	0.028000	0.028000	100	400	yes	hole.028
2	0.045000	0.045000	0.045000	100	400	yes	hole.045
3	0.180000	0.180000	0.180000	100	400	yes	hole.18

## Simulating Drilling

You can simulate the tool paths for a drill file, as shown in [Figure 4-3](#). The path for each tool in the file displays on a different layer so that you can easily distinguish paths. The simulation also displays the number of holes drilled with each tool and the drill head travel. The dashed lines in the tool paths are returns to the home position.



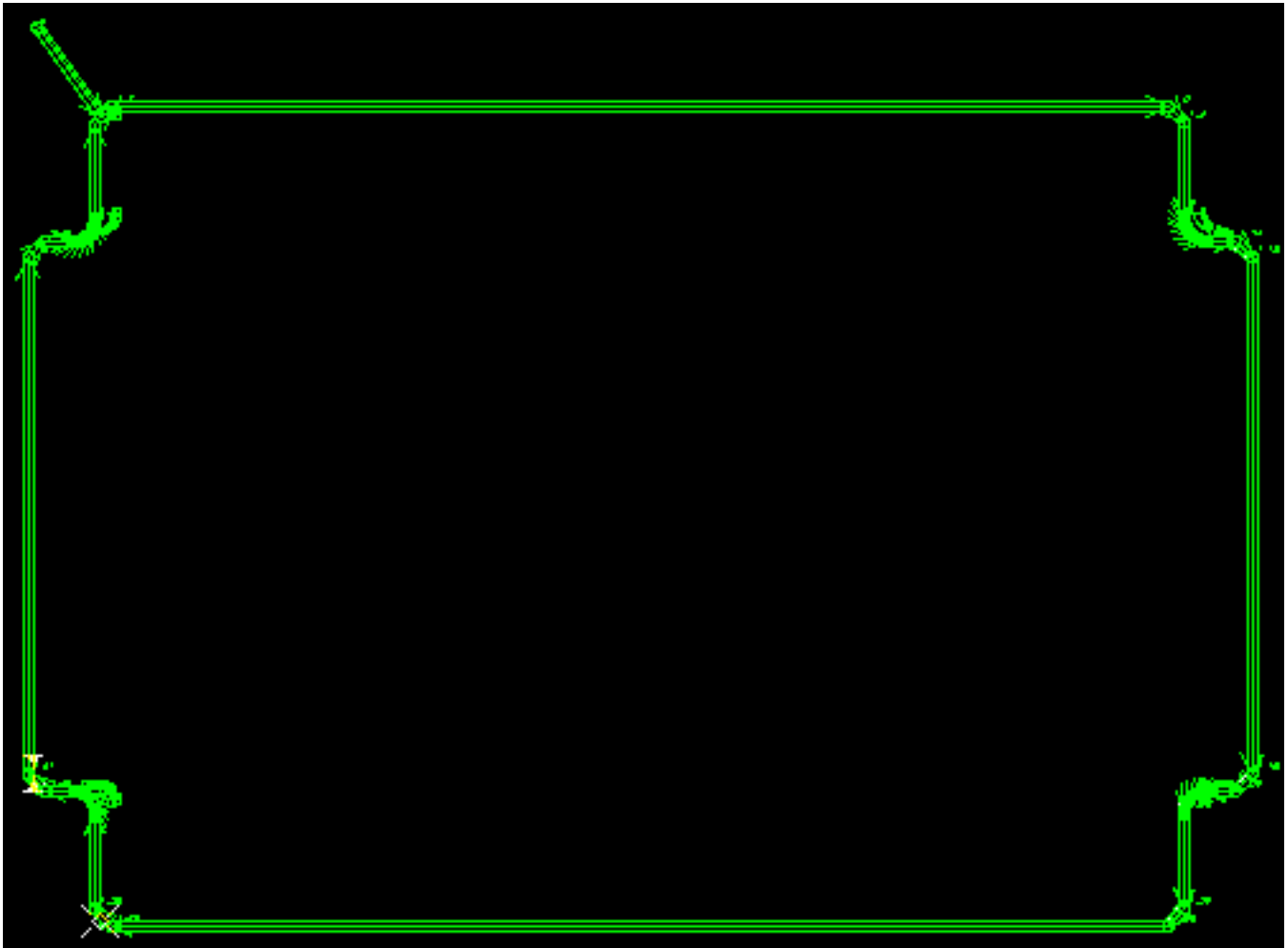
**Figure 4-3. Example Simulated Drill Path**

## Defining the Milling Tool Paths

Follow this procedure to define the paths used by the milling machine. For a more detailed description consult the section “Defining the Milling Tool Paths” in the *Using PCB Design Tools* manual.

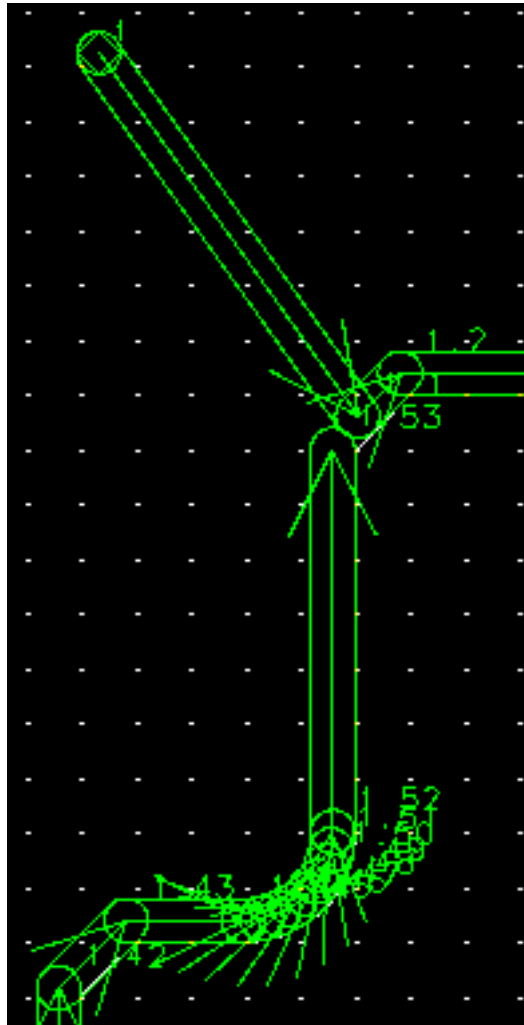
1. Set the edit layer to the layer you want to mill. The Milling layer is recommended, although you can apply milling data to any layer. Turn on the layers you want to view. Turn on the layer(s) with data you want to follow. For example, to mill the board outline, turn on the Board\_outline layer.
2. Check the characteristics of the current milling tool bit by choosing the [Top Menu] **Milling > Change Tool** popup menu item from the board geometry edit window.
3. Apply the milling plunge and path points, either interactively or automatically.

An example is shown in [Figure 4-4](#) and [Figure 4-5](#).



**Figure 4-4. Example of an Automatic Milling Path**

- Choose the **[Top Menu] Milling > Start Tool Path** popup menu. This function sets the Select mouse button for entering the milling path. The first click enters the plunge point, and subsequent clicks enter path points. The plunge and path points are identified by sequence numbers.
- You can automatically create a milling path that traces the shape of an existing object, such as a board outline, by choosing **[Top Menu] Milling > Start Tool Path** and specifying a point on the object you want to trace. When the Add Milling Tool Path prompt bars appears, toggle the stepper button to *automatic*. Use the Select mouse button to select a point on the object.



**Figure 4-5. Close-up of the Milling Path**

4. While milling, you can undo the latest path or plunge entries by pressing the Backspace key. Press Backspace more than once to step backwards until you have deleted everything, including the plunge point. The Backspace key deletes milling points only when the Add Milling Tool Path prompt bar is active.

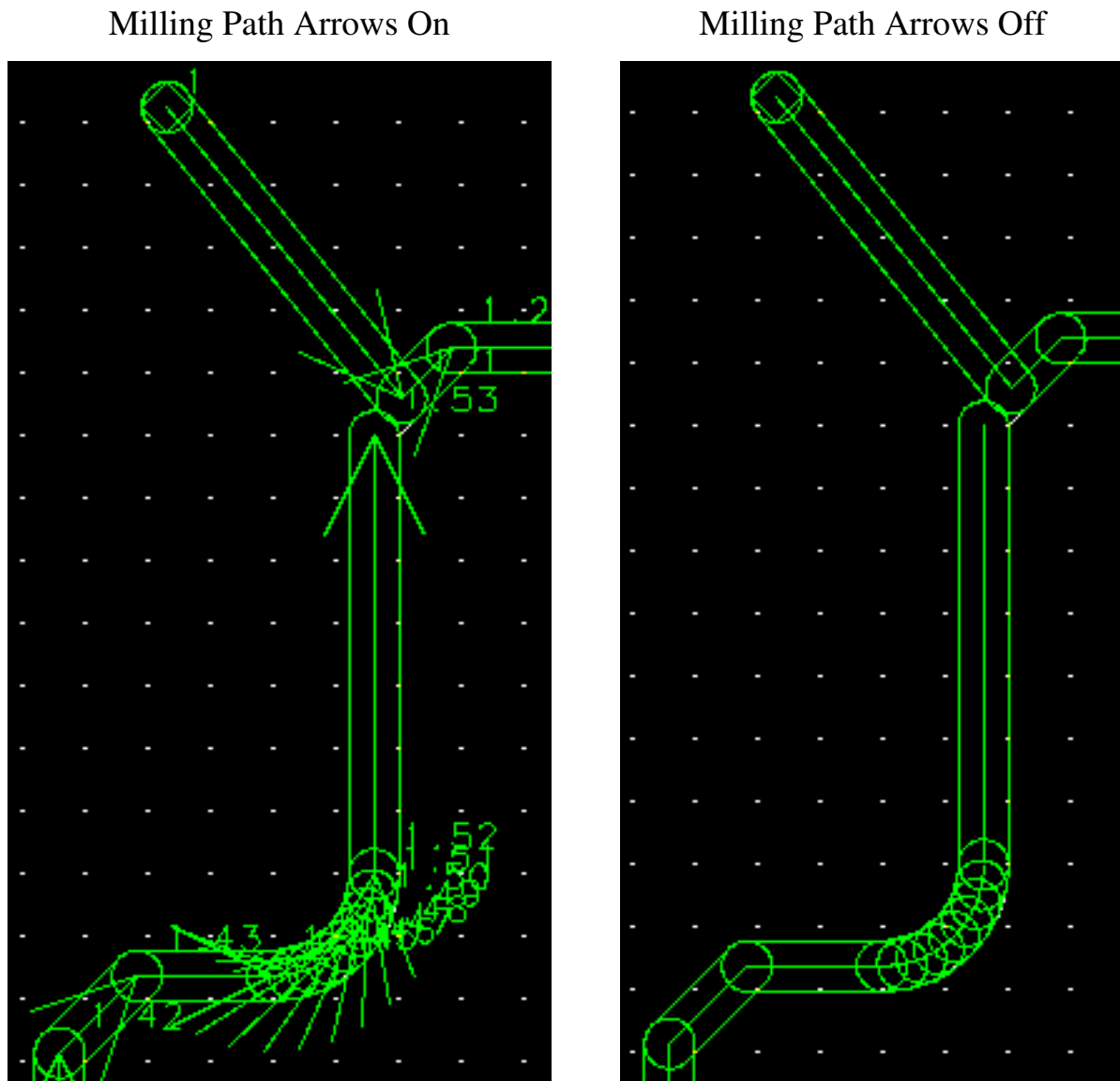
## Changing the Milling Tool Size

You can select a completed milling path and change the size of the tool used to create the path. FabLink adjusts the milling path based on the new tool size chosen.

## Hide Milling Arrows and Order Numbers

By default, FabLink displays milling paths with direction arrows and order numbers. When you create drawings for your design, you might want to indicate the direction and order of the milling path without using these arrows and order numbers. Refer to [Figure 4-6](#) to see an example of milling paths with milling arrows and order numbers hidden and not hidden. To turn the order numbers and arrows off, choose **View > Milling Path Arrows Off**. To turn the arrows on, choose **View > Milling Path Arrows On**. Normally, the arrows and order numbers are visible.





**Figure 4-6. Examples of Milling Paths**

## Opening Drill/Mill Data

You can view drill and milling data on any layer that you specify. By specifying the layer on which to view the data, you can overlay the images in one Edit window and still distinguish among the images.

To modify viewed drill or milling data, you can make modifications on any layer. When saving the data, you can specify the layers you modified, plus the layer on which you viewed the original data. The output file includes the data on all of the specified layers.



*When you edit and save drill data, only the output data is altered. The PCB design data used to create this output (for example, your geoms design object and drill table) is not changed. You are responsible for the additions and deletions, because the system does not check the altered drill data for design rule violations.*

## Lab Exercise

In this lab exercise, you create two additional types of manufacturing data: numerical control (NC) drill and milling data. The procedure for creating a drill table is similar to the procedure you used in the previous lab exercise to create an aperture table. You use the drill table to create the drill data for your panel. Then, you simulate the drill data and observe the results. Finally, you generate tool paths to route the board from the panel and create the milling data.

Upon completion of this lab exercise you are able to:

- Create a drill table.
- Create drill data for the panel.
- Simulate the drill data.
- Set the characteristics for the milling tool.
- Create a milling path automatically.
- Create milling data.

Turn to Module 7—Lab 4: “Creating Drilling and Milling Data.”



# Lab 4

## Creating Drilling and Milling Data

### Introduction

In this lab exercise, you create two additional types of manufacturing data: numerical control (NC) drill and milling data. Creating a drill table is similar to the procedure used in Lab 3 to create an aperture table. In this lab you use the drill table to create drill data for your board. You then simulate the drill data and observe the results. Finally, you generate tool paths to route the board from the panel and create the milling data.

Upon completion of this lab exercise you are able to:

- Create a drill table.
- Create drill data for the board.
- Simulate the drill data.
- Set the characteristics for the milling tool.
- Create a milling path automatically.
- Create milling data.

## Procedure

In this lab exercise you use FabLink to create NC drill and milling data.

## Preparation for Lab

1. Invoke the Design Manager. Double click on the FabLink icon in the Design Manager Tools window. In the navigator dialog box, select the `your_path/training/board_new/mod7/sig_az` design, then press **OK**.
2. Close the report window.

## Creating Drill Data

Like the artwork data you simulated earlier in this module, in this lab you view a simulation of the drill data. You begin by using the standard technique of filling the drill table.

1. Choose **[Top Menu] Drill > Change Drill Table > Fill Drill Table...** In the Fill Drill Table dialog box, choose **OK** without making any changes.

A report window displays.

2. Close the report window.

Next, you look at the drill table.

3. Choose **Report > Drill Table...**, then choose the following values in the dialog box and press **OK**.

**Include Drill Format**  
**Display Report**

A Notepad window lists the drill position assignments.

4. Read the window contents. After viewing the drill assignments, close the Notepad window.

Now you are ready to create the drill data.

5. Choose **[Top Menu] Drill > Create Drill Data...**, enter the following values into the dialog box, and press **OK**.

Drill File Format:	<b>Excellon</b>
Generate for:	<b>Step &amp; Repeat Panel</b>
Drill Character Set:	<b>ASCII</b>
Mirror Output:	<b>No</b>
Drill Hole Types:	<b>Plated Holes</b>
<b>Generate All Layer Pairs</b>	

For each layer pair, a report window displays listing the total vias found and other information. Only the last report window remains visible.

6. Close the report window(s).

Next you simulate the drill data.

7. Choose **[Top Menu] Drill > Simulate Drill Data...**, and choose **Select** in the dialog box. Select **drill\_1\_4** from the list, then press **OK**.

A new edit window displays, showing the simulated drill data of the drill\_1\_4 path. A block of text below the drill path contains drilling information. If you want, you can change the view to see the text better. If you have time, try viewing some of the other drill paths by repeating this step and selecting a different drill path from the dialog box.

8. Close the edit window containing the drill simulation.

The BO\$Board edit window, showing the board, is the only edit window visible.

## Creating Milling Data

In this procedure, you explore two options for creating and changing milling data quickly. The first option allows you to automatically generate a milling path for a board feature such as the board outline. The second option allows you to change the tool size used to mill a path without regenerating the path.

1. Set up the edit layer to Milling.
2. Set up the view layers to view only Board\_outline and Milling.

FabLink checks features on visible layers when generating a milling path, so only the Board\_outline and Milling layers is visible. Next, you set up the milling tool size.

3. Choose **[Top Menu] Milling > Change Tool...**, enter the following values in the dialog box, and press **OK**.

Tool Diameter:	<b>.062</b>
Tool Compensation:	<b>Left</b>
Plunge at:	<b>Absolute Location</b>
Digitize Point at:	<b>Nearest Vertex</b>

You are now ready to create the milling path.

4. Choose **[Top Menu] Milling > Start Tool Path:**.

A prompt bar displays, and you are prompted for a location. This location marks the milling tool's plunge point.

5. Click the Select mouse button with the cursor at a point about 0.5 inches above the upper-left corner of the board outline.

The plunge point is defined, and you can see a small green (default milling layer color) dot showing the plunge point. If you move the cursor, you can see a ghost image line connecting the plunge point to the cursor.



6. In the prompt bar, click on the stepper arrows so that **automatic** displays in the first prompt. Also specify to **repeat**. Finally, Tab to the location prompt. Click the Select mouse button with the cursor on or very near the upper-left corner of the board outline.

The complete tool path around the board outline is created automatically. If the tool path stops at some point around the board before completing the path, such as at a chamfer or arc, it might be because of a very small (invisible) space between a line and an arc, or between a line and a chamfer.

To continue the milling path, you first verify that the Add Milling Tool Path prompt bar displays, then you click on the next segment of geometry after the vertex where the milling path stopped. The milling path continues from that point either until the path around the board is complete, or until the milling path stops at another vertex. If it stops again, you can click on the next segment until the milling path is complete.

To create different milling paths, you can select the milling path and delete it. (Selecting near the plunge point is convenient so that you do not accidentally select the board outline.) You can then create a new milling path, even using a different offset or tool size.

7. Cancel the prompt bar.

It is possible to change the tool size after the milling path is created.

8. Choose **[Top Menu] Milling > Change Tool Size:**, and when the prompt bar displays, select the milling path by placing the cursor on the milling path near the plunge point and clicking the Select mouse button (if you click near the board outline, you might also select the board outline accidentally). In the **New Width** field of the prompt bar, enter **.05** and **OK** the prompt bar or press the Return key.

After a moment, the milling path is redrawn to the new size and unselected automatically.

After the milling path is complete, you need to complete the process of creating the milling data by filling in the milling table and creating the milling data. You do this in the next few steps.

9. Choose **[Top Menu] Milling > Change Milling Table > Fill Milling Table...**, leave the settings in the dialog box at the default values, and press **OK**.

A report window displays.

10. Close the report window.

11. Choose **[Top Menu] Milling > Create Milling Data...**, leave the dialog box settings at the default values, and press **OK**.

A report window displays, showing the pathname to the milling data file FabLink created. You do not have to specify a filename in the Create Milling Data dialog box, unless you want to create more than one milling file for a design.

12. Close the report window.

13. Choose **[Top Menu] Milling > Open Milling Data...**, leave the dialog box settings at the default values, and press **OK**.

A new edit window, named AR\$milling, showing the milling data displays.

You have now created most of the data required to manufacture the lab design board. Your company's manufacturing processes might invalidate some of the assumptions made for this lab, but in general, the basic procedures you completed in this lab are similar for many different processes.

14. Close the FabLink session and save the changes to your design.

Congratulations! You have completed the "Creating Drilling and Milling Data" lab exercise. Continue with Lesson 5: "Creating Fabrication and Assembly Drawings."

# Lesson 5

## Creating Fabrication and Assembly Drawings

Drawings and reports are the final pieces required to manufacture a printed circuit board.

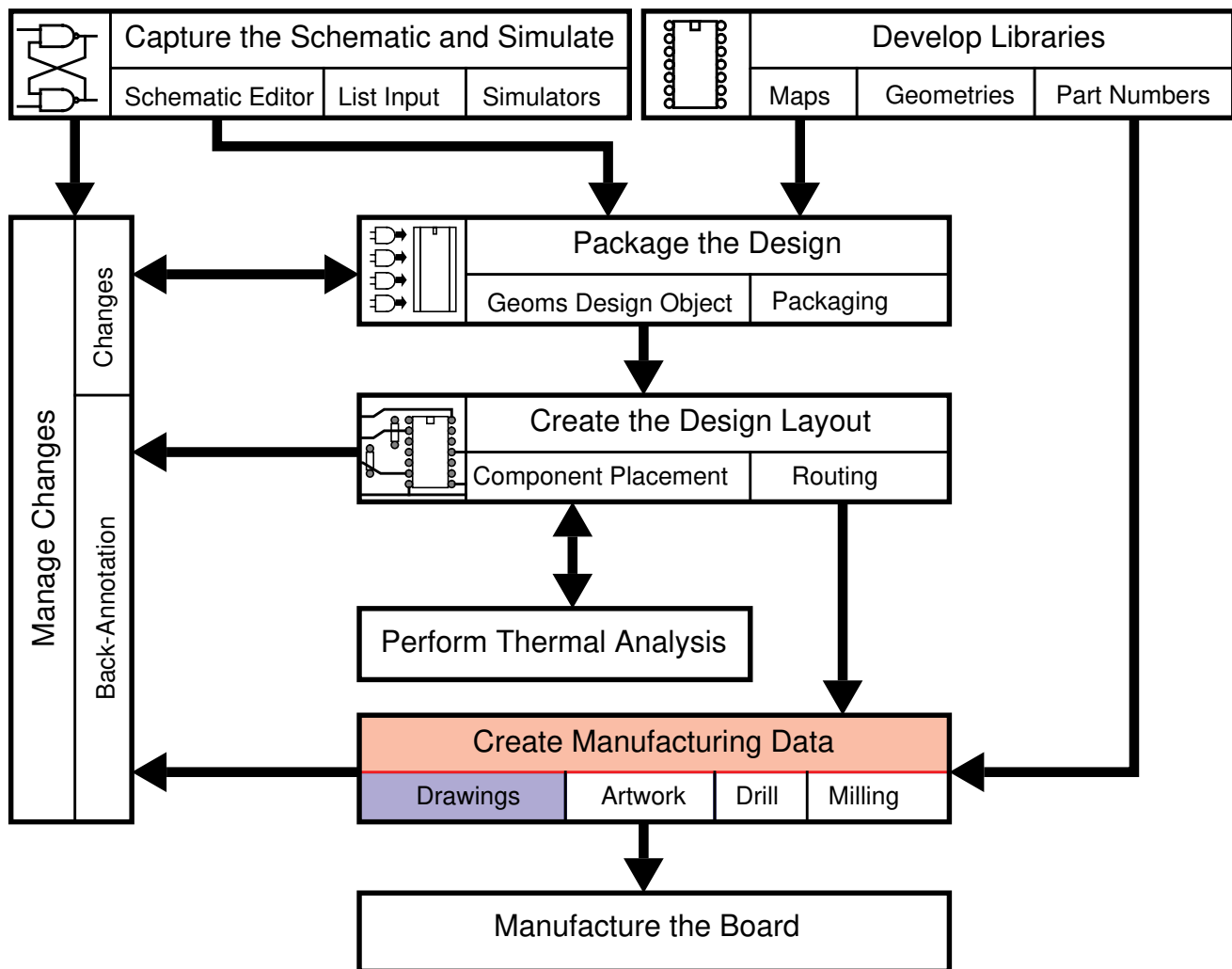


Figure 5-1. PCB Design Process

## Objectives

After completing this module, you are able to describe:

- How to create a fabrication drawing
- How to create an assembly drawing
- Basic drafting functions
- Basic dimensioning functions
- The Aperture Table and Drill Table reports
- The process of customizing the Drill Schedule and Bill of Materials

## Drafting and Reports

Generating artwork, drilling, and milling data does not usually complete the required data set for manufacturing a printed circuit board. A fabrication drawing is required to specify the characteristics (dimensions, material, and processes used for manufacturing) of the printed circuit board. Another drawing is required to describe how the board and other parts are assembled to produce a complete circuit board. This drawing is called an assembly drawing.

A listing of the aperture table and drill magazine assignments is usually needed either for the manufacturing facility or for the designer's reference. You might also want to include on your drawings a drill schedule, containing drill size and quantity information, and a bill of materials.

In this lesson, you examine how to create each of these documents.

## Creating a Fabrication Drawing

A typical fabrication drawing depicts the dimensional configuration of the printed circuit board, size and location of drill holes, material and process specifications, and notes and other information necessary to fabricate the board.

Create a fabrication drawing as follows:

1. Choose **Geometries > Create Geometry > Drawing**, and fill in the dialog box. The system generates the fabrication drawing sheet border when you press **OK**.
2. Add a board geometry to the drawing by choosing the **[Top Menu] Drawing > Add Board** popup menu item. When you click the Select mouse button, the board is added at the specified location.
3. Drawing information is generally placed on the Drawing\_1 layer. Set your edit layer to Drawing\_1 by choosing **Setup > Edit Layer**. Any editing changes or additions are applied to the Drawing\_1 layer. Be sure to make the layers visible before you begin drawing.
4. Unless it is already loaded, create or restore a drill table. Refer to section ["Defining and Saving the Drill Table" on page 4-3](#) for more information.
5. Read the drill symbols into the session as library geometries from the Mentor Graphics-supplied directory *drill\_symbols* or from any of your own custom geometry libraries.
6. Create and display the drill schedule by choosing **Geometries > Create Geometry > Drill Schedule**. Use the dialog box to customize the drill schedule for you design.
7. Choose **View > Drill Symbols On**. This menu item displays symbols for plated and unplated through-pin drill holes, tooling holes, and vias.
8. Choose the **[Top Menu] Drawing > Add Format Block > Add Drill Schedule** popup menu item. Point to where you want the top-left corner of the schedule and click the Select mouse button.
9. Add text, pointers, dimensions, a board side view, and feature control blocks to the fabrication drawing as required by your drafting standards.

## Customized Drill Schedules

When you create your drill schedule using **Geometries > Create Geometry > Drill Schedule**, and the dialog box shown in [Figure 5-2](#), you can customize the drill schedule by specifying which columns to include, the order of the columns, and the column titles.

You can also define columns for tool position and comments and specify a title for the drill schedule. If you use the default setting Layer Schedule, you see a list of layers for your design, such as that shown in [Figure 5-2](#). The resulting drill schedule is for the single layer you had selected in the list, and looks like the drill schedule shown in [Figure 5-3](#).

If you choose Master Schedule, the list of layers is removed from the dialog box, and the resulting drill schedule is for all layers of the design. It then appears like the drill schedule shown in [Figure 5-4](#).

Images of the drill symbols are specified by entering geometry names for the symbols in the drill table for each drill used in the design.

**Create Drill Layers Schedule**

---

Create Drill for  
☒ Board ☐ Panel

Title for Board Drill Schedule

---

☐ Master Schedule ☒ Layer Schedule

From Layer	To Layer
Trace_Layer_1	VCC
Trace_Layer_2	Trace_Layer_3
ground	Trace_Layer_4
Trace_Layer_1	POS15V
NEG15V	Trace_Layer_4

---

**Note: Columns at Position '0' will not be included in the Schedule.**

Drill Symbol	Position <input type="text" value="1"/>	Title <input type="text" value="SYMBOL"/>
Drill Size	Position <input type="text" value="2"/>	Title <input type="text" value="SIZE"/>
Count	Position <input type="text" value="3"/>	Title <input type="text" value="QTY"/>
Plated	Position <input type="text" value="4"/>	Title <input type="text" value="PLATING"/>
Tolerance	Position <input type="text" value="0"/>	Title <input type="text" value="Min/Max"/>
Tool	Position <input type="text" value="0"/>	Title <input type="text" value="TOOL"/>
Comment	Position <input type="text" value="0"/>	Title <input type="text" value="COMMENT"/>

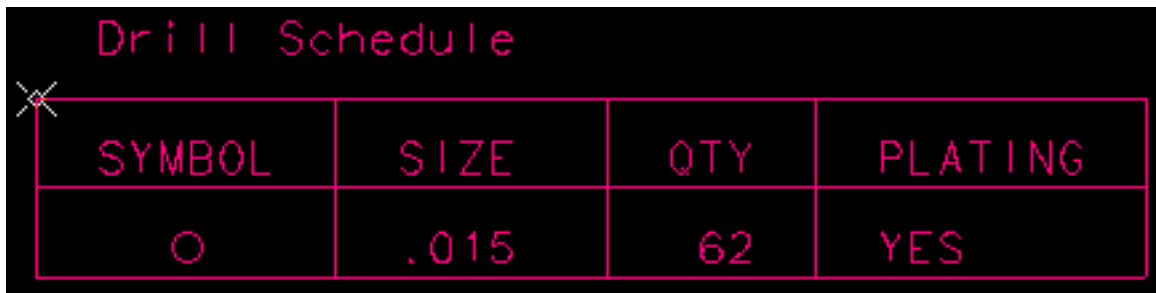
---

Plated 'Yes' Title <input type="text" value="YES"/>	<input type="checkbox"/> Include Lead Zeros
Plated 'No' Title <input type="text" value="NO"/>	

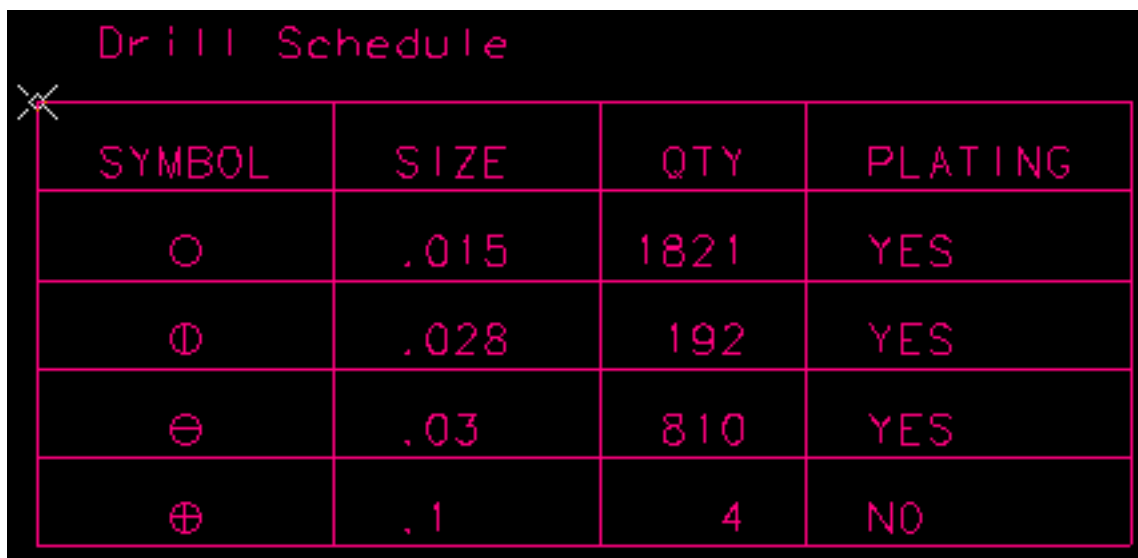
---

Units ☒ Inches ☐ CM ☐ MM ☐ Mils ☐ Tenth Mils

Figure 5-2. Create Drill Schedule Dialog Box



SYMBOL	SIZE	QTY	PLATING
○	.015	62	YES

**Figure 5-3. Drill Schedule for a Single Layer**

SYMBOL	SIZE	QTY	PLATING
○	.015	1821	YES
⊕	.028	192	YES
⊖	.03	810	YES
⊕	.1	4	NO

**Figure 5-4. Master Drill Schedule**



## Creating an Assembly Drawing

An assembly drawing typically describes the assembled printed circuit board with all electrical components and mechanical parts contained on a particular assembly. It depicts the circuit board components, reference designators, markings, and test specifications. It might include a parts list or bill of materials. The following steps describe the process of creating an assembly drawing:

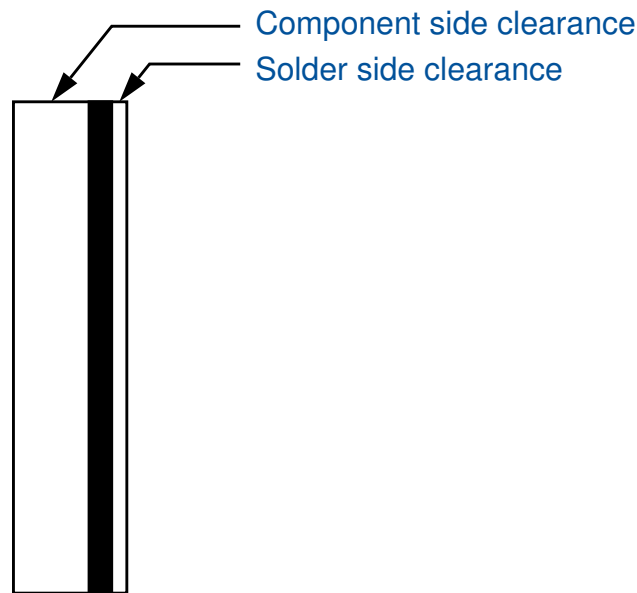
1. Choose **Geometries > Create Geometry > Drawing**. Fill in the dialog box according to how you want the assembly drawing to look. The system generates the assembly drawing sheet border when you press **OK**.
2. To add a board geometry to the drawing, choose the [**Top Menu**] **Drawing > Add Board** popup menu item from the board geometry edit window. When you click the Select mouse button, the board is added at the specified location.
3. Add text, pointers, dimensions, a board side view, and parts list to the assembly drawing, as required by your drafting standards.

## Basic Drafting

FabLink also has a variety of features that help companies comply with military standard drafting practices (DOD-1000). When you are adding dimensions to drawings, there are a variety of options that also assist in complying with military standards (ANSI Y14.5 as referenced in DOD-1000). The Mentor Graphics-supplied parts lists, title blocks, revision blocks, and various sheet sizes that conform to military specifications (ANSI Y14.1 as referenced in DOD-1000) are located in the `$MGC_PCBPARTS/pcb_geoms/drawing_formats.ascii` directory.

## Adding the Board's Side View

When you create a drawing, you can use FabLink to add a side view of the board to a drawing to show the board thickness and top and bottom component clearance heights, as shown in [Figure 5-5](#).



**Figure 5-5. Example Side View Drawing**

## Adding Dimensions

You can quickly add dimensions for straight lines, drill holes, angles, and other features to any geometry, including circuit board drawings. The distances and angles of the circuit board and other geometries are measured automatically. Dimensions you add are attached to the geometry that you are dimensioning; they are not added to the drawing geometry. The major steps in dimensioning are:

1. Set up the dimensioning style.
2. Select the points you want to dimension.
3. Add dimensions.
4. Change dimensions.

## Setting Dimensioning Style

Before adding dimensions, you can set the dimension style by choosing **Setup > Dimension**. The options in the dialog box include:

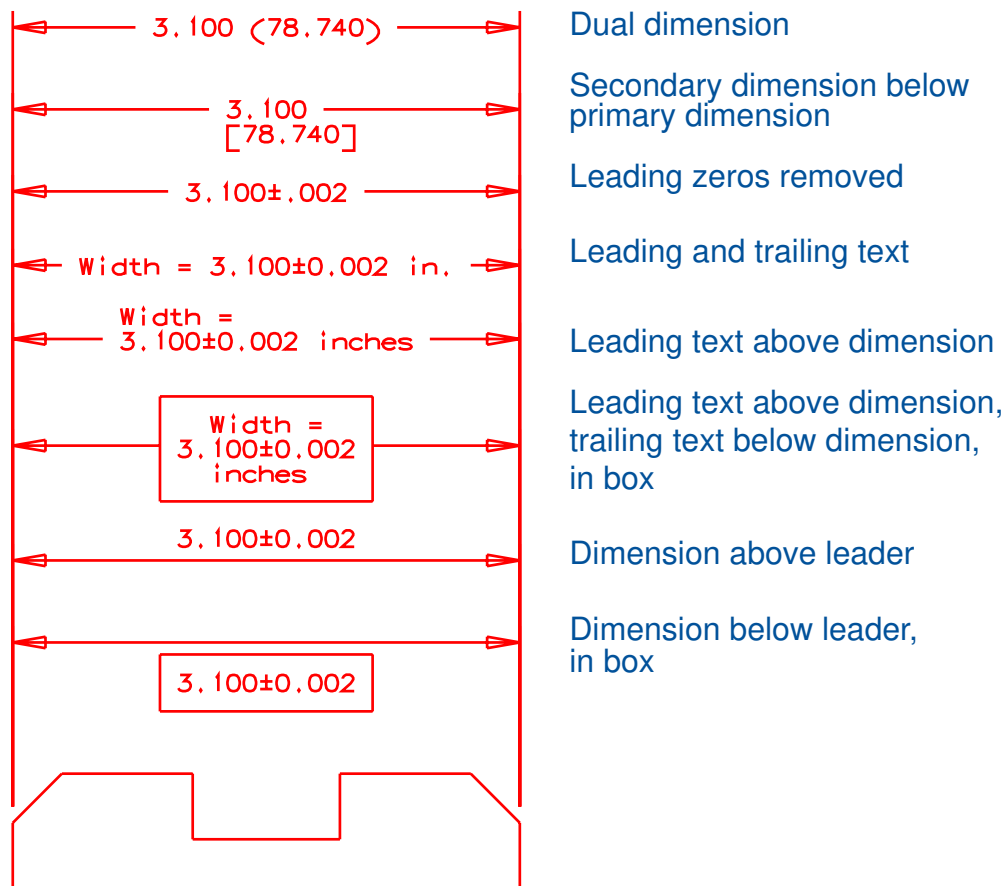
- **Arrowhead**—Below the Arrowhead label, choose different arrowhead styles for each end of a dimension line.
- **Extension Line**—Choose an extension line style of Centerline, Dashed, Solid, or None for the first and second dimension points.
- **Leader Segment**—Choose to create leader segments automatically, to place them inside or outside extension lines, or to use no leader segments.
- **Extension Line/Leader**—Enter the line width for extension and leader lines.
- **Tolerance**—Specify the desired tolerance style: Bilateral, Unilateral, Limit, or none.
- **Precision**—Choose the number of digits of precision.
- **Dimension Text**—Under the Dimension Text label, choose various dimension text format parameters such as font, height, position, and orientation of the text display.
- **Dual Units Format**—Choose whether you want to display the dimension with more than one unit of measure.

The following options are used only if you choose the Clearance Checking option when you add a horizontal or vertical dimension:

- **Leader-to-Leader Minimum Clearance**—Sets minimum spacing between the leaders of adjacent dimensions. Default value is .38 inches or 10 mm.
- **Leader-to-Feature Minimum Clearance**—Sets minimum spacing between a dimension leader and the dimensioned feature. Default value is .5 inches or 12 mm.

## Dimensioning Styles

You can display dimensions with dual values (primary and secondary units) and with leading zeros removed. You can position leading and trailing dimension text above or below the dimension value. You can even enclose dimensions in rectangular boxes and place linear dimensions on either side of the leader line. Figure 5-6 shows examples of dimension styles.



**Figure 5-6. Dimension Style Examples**

Text

You can enclose text with any of eight outlines: square, rectangle, circle, triangle, tear, diamond, flag, or banner. By default, outlines are sized to encompass the text. You can also control the height and width of the outlines. In addition, the iges1003 character font is available for adding feature control symbols to drawings. You can mix outlines and fonts to create a variety of text objects. Figure 5-7 shows examples.

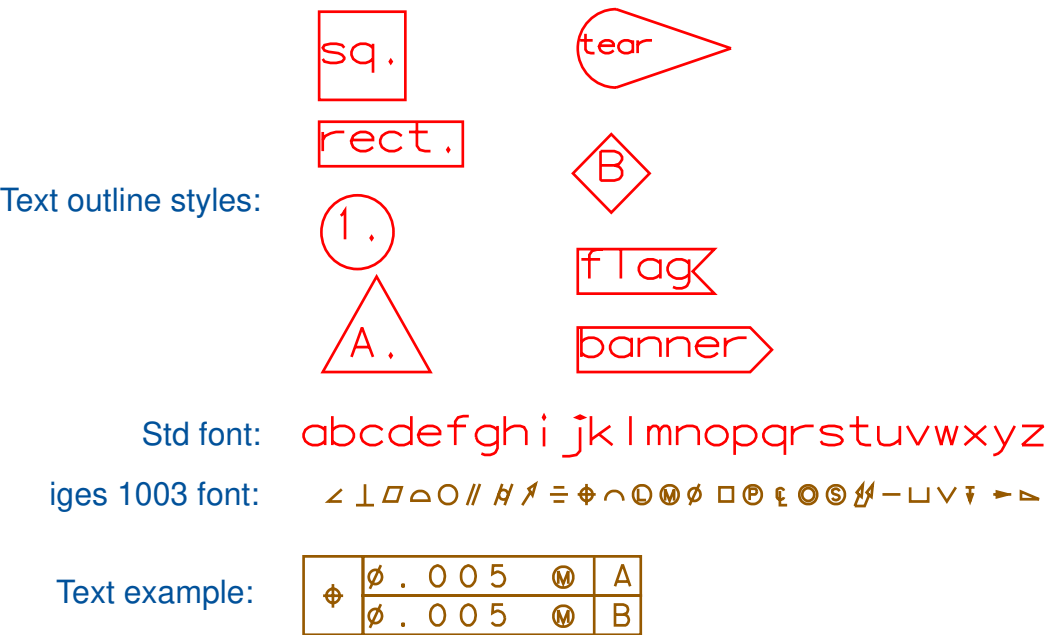


Figure 5-7. Text Examples

## Selecting Points to Dimension

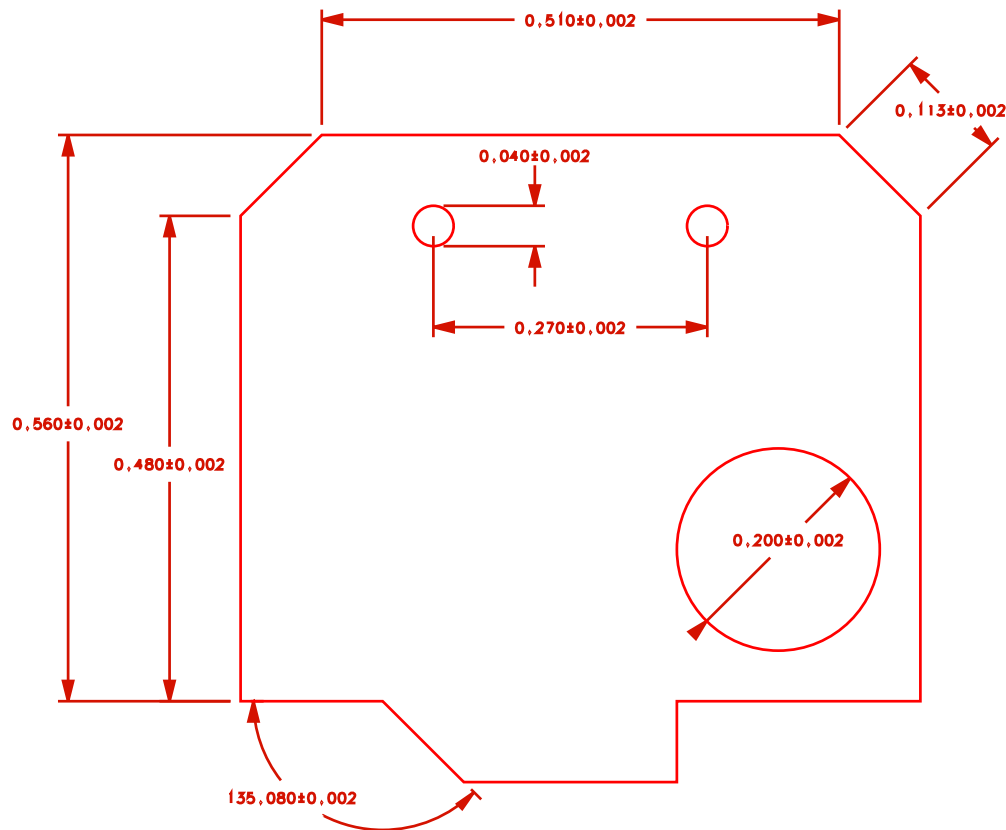
When adding dimensions, you can choose what kind of points are selected for the first and second points of the dimension. To set your point selection mode, first specify the type of dimension you want to add using the appropriate Dimension menu item, then press the Options button in the prompt bar.

For example, for a horizontal dimension choose the **[Top Menu] Dimension > Add Horizontal Dimension** popup menu item, then press the Options button in the prompt bar. A Selection Mode options dialog box lists your choices. The following options are available:

- **Absolute**—selects the cursor location.
- **Bottom**—selects the bottom center of an element.
- **Bottom Left**—selects the bottom left of an element.
- **Bottom Right**—selects the bottom right of an element.
- **Center**—selects the center of an element, for example the center of a circle.
- **Left**—selects the left center of an element.
- **Origin**—selects the origin of an element.
- **Right**—selects the right center of an element.
- **Top**—selects the top center of an element.
- **Top Left**—selects the top left of an element.
- **Top Right**—selects the top right of an element.
- **Vertex**—selects a vertex of a drawing or geometry.

## Adding Dimensions

Add dimensions, such as those shown in [Figure 5-8](#), as follows:



**Figure 5-8. Dimension Examples**

1. Choose the [Top Menu] Dimension > Add Horizontal Dimension popup menu item, or:

[Top Menu] Dimension > Add Vertical Dimension

or

[Top Menu] Dimension > Add Parallel Dimension

or

[Top Menu] Dimension > Add Angular Dimension

or

[Top Menu] Dimension > Add Radial Dimension

2. In the prompt bar, press the Options button to check your point selection mode. Change the type of point selected for the first and/or second dimension points, if wanted. If adding a vertical or horizontal dimension, you can also check leader-to-leader and leader-to-feature clearances. To add several dimensions, press the Repeat check button. After reviewing your choices, press **OK**.

The dialog box closes. A prompt bar prompts for dimension locations.

3. Click with the cursor on the first dimension point.

An icon and the number 1 appears on this location.

4. Click with the cursor on the second dimension point.

An icon and the number 2 appears on this location. A ghost image of the dimension appears.

5. Use the mouse to position the ghost image of the dimension and control the length of the extension line.

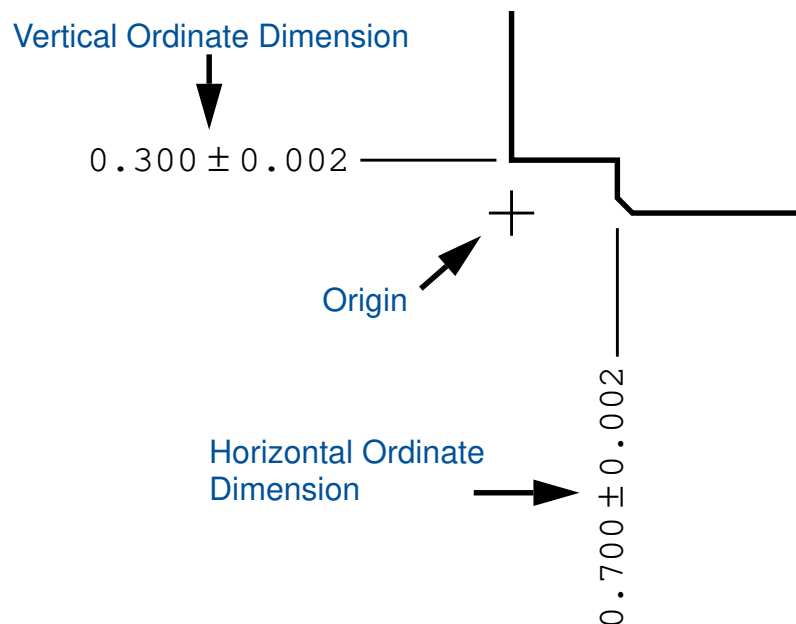
If the dimension style specifies *Auto* leader segments, the cursor position controls whether the leader segments are inside or outside the dimension.

6. When satisfied with the dimension location, click again. The dimension is added.



## Adding Ordinate Dimensions

In addition to linear, angular, and radial dimensions, you can add ordinal dimensions, both horizontal and vertical, as shown in [Figure 5-9](#). Ordinal dimensions display at the end of their extension lines, with no leaders to the datum point.



**Figure 5-9. Ordinal Dimension Examples**

To add a horizontal or vertical ordinate dimension:

1. Choose **[Top Menu] Dimension > Set Ordinate Origin**.
2. In the prompt bar, press Options to check your point selection mode for the origin.

The prompt bar prompts for the dimension's origin.

3. Click on the origin's position.

An icon and the number *I* appears on this location.

4. Choose either:  
[Top Menu] **Dimension > Add Horizontal Ordinate Dimension**  
or  
[Top Menu] **Dimension > Add Vertical Ordinate Dimension**
5. Next, supply the location of second point of the dimension from the Add Vertical/Horizontal Ordinate prompt bar.  
  
If wanted, you can first change the type of point selected for the dimension using the Options button.
6. Click on the position and then use the mouse to control the length of the extension line.

## Manufacturing Reports

FabLink supports many features for generating manufacturing reports. These features are described briefly in this section. For more information on these reports, refer to the *Using PCB Design Tools* manual.

### The Aperture Table Report

When you view the apertures or write out the apertures in report form, you can specify that the column containing the X dimensions of the apertures comes before the column of Y dimensions. This allows you to view the aperture dimensions in the order that you are accustomed to viewing them. [Table 5-1](#) shows a standard aperture table report.

**Table 5-1. Standard Aperture Table Report**

Aperture	D_Code	Shape	Type	Power	Height(Y)/ Diameter	Width(X)
1	10	circle	trace		0.010000	0.000000
2	11	circle	flash		0.028000	0.000000
3	12	circle	trace		0.012000	0.000000

Table 5-2 shows an aperture table report with X and Y reversed.

**Table 5-2. Aperture Table Report With X-Y Reversed**

Aperture	D_Code	Shape	Type	Power	Height(Y)/ Diameter	Width(X)
1	10	circle	trace		0.000000	0.010000
2	11	circle	flash		0.000000	0.028000
3	12	circle	trace		0.000000	0.012000

## The Drill/Mill Table

When you fill the drill/milling table from the design data, the table is sorted by tool size from smallest to largest, as shown in Table 5-3.

**Table 5-3. Example Drill Table**

Drill Position	Drill Size	Upper Bound	Lower Bound	Feed Rate	Speed Rate	Plated	Symbol
1	0.028000	0.028000	0.028000	100	400	yes	hole.028
2	0.045000	0.045000	0.045000	100	400	yes	hole.045
3	0.180000	0.180000	0.180000	100	400	yes	hole.18

## Customized Bill of Materials

Choose **Report > Bill of Materials...** to customize a bill of materials. You specify which columns to include, the order of the columns, and the column titles, as shown in [Figure 5-10](#). Specifying a zero (0) in the Position box removes the field from the bill of materials report. The description field can also include property value information, such as the value and tolerance for resistors. You can also define a column for comments. The option to customize the bill of materials is available in both FabLink and PACKAGE.

	Position	Title
Item Number	1	ITEM_NUMBER
Part Name	2	COMPANY PART NO.
Geometry	3	GEOMETRY
Count	4	COUNT
Description	5	DESCRIPTION
Reference Name	6	REFERENCE
Comment	0	COMMENT

Figure 5-10. Portion of the Bill of Materials Dialog Box

## Lab Exercise

In this lab exercise, you complete the drawing and bill of materials necessary to fabricate the lab design board. You complete a detail of the upper-left corner and tooling hole. You then create a fabrication drawing for the design and add the completed detail to the drawing. You also create an assembly drawing for the board. The last task is to create a bill of materials with customized column headers and column ordering.

Upon completion of this lab exercise you are able to:

- Complete a drawing detail.
- Create a fabrication drawing.
- Create an assembly drawing.
- Create a customized bill of materials.

Turn to Module 7—Lab 5: “Creating Fabrication and Assembly Drawings.”



# Lab 5

## Creating Fabrication and Assembly Drawings

### Introduction

In this lab exercise, you complete the drawings and bill of materials necessary to fabricate the lab design board. You complete a detail of the upper-left corner and tooling hole. You then create a fabrication drawing for the design and add the completed detail to the drawing. You also create an assembly drawing for the board. The last task is to create a bill of materials with customized column headers and column ordering.

Upon completion of this lab exercise you are able to:

- Complete a drawing detail.
- Create a fabrication drawing.
- Create an assembly drawing.
- Create a customized bill of materials.

## Procedure

Use FabLink to create drawings.

## Preparation for Lab

1. After invoking the Design Manager, double click on the FabLink icon in the Design Manager Tools window. In the navigator dialog box, select the `your_path/training/board_new/mod7/sig_az` design, then press **OK**.
2. Close the report window.

The view you see of the board might show only the milling layer. Later, you reset your edit and view layers.

## Completing a Detail

In your User geometry library is the beginning of a board corner detail. In this section of the lab, you complete the dimensioning of the detail, including a feature control block. You also place a label on the detail.

1. Read into the FabLink session the geometry named *detail\_a* from the *mgc.trng.drawings* User library.

If you do not have a *mgc.trng.drawings* library, refer to section ["Creating a Link to a Geometry Library" on page 3-16](#) for instructions on how to create a link to this library. You can also ask your instructor for help.

A corner detail of the board design is read into a new edit window. You add dimensions to complete the detail in the next steps. A report window displays over the edit window.

2. Close the report window.
3. Activate the `E$detail_a` edit window and view all of the detail drawing.
4. Verify that the edit layer is set to `Drawing_1`.



5. Choose **View > Layers**, and verify that Drawing\_1 layer is both visible and selectable (there is a V and an S next to it).
6. Set the grid to .05, with an interval of 2.
7. Set the snap direction to any angle.
8. Choose **Setup > Text**, enter the information from Table 5-4 in the dialog box, and press **OK**.

**Table 5-4. Setup Text Dialog Box Settings**

Height: <b>.125</b>	Auto Mirror Text
Orientation: <b>0</b>	Font: <b>iges1003</b>
Aspect Ratio: <b>1</b>	Force Right Reading
Stroke Width: <b>.01</b>	Justification: <b>Bottom Left</b>
Line Spacing: <b>1.6</b>	Box Style: <b>None</b>

9. Choose **Setup > Pointer**, enter the information from Table 5-5 in the dialog box, and press **OK**.

**Table 5-5. Setup Pointer Dialog Box Settings**

Arrow	Text
Height: <b>.042</b>	Height: <b>.15</b>
Length: <b>.125</b>	Aspect Ratio: <b>1</b>
Style: <b>Solid Arrow</b>	Stroke Width: <b>.01</b>
Line Width: <b>0</b>	Font: <b>iges1003</b>
Gap Value: <b>.1</b>	Box Style: <b>Triangle</b>
Line Style: <b>Solid</b>	Justification: <b>Center Left</b>

10. Choose **Setup > Dimension**, enter the information from [Table 5-6](#) in the dialog box, and press **OK**.

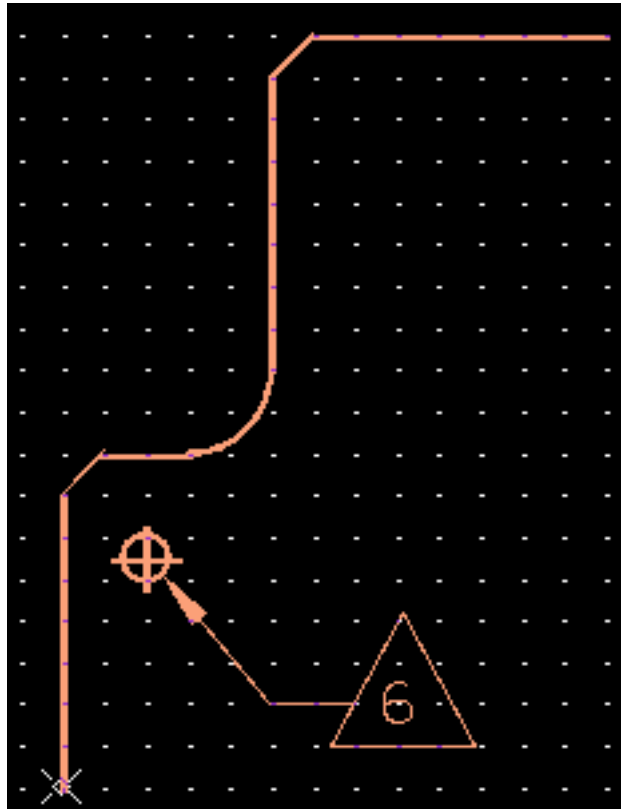
**Table 5-6. Setup Dimension Dialog Box Settings**

Arrowhead -Arrow1 Style: <b>Solid Arrow</b> Height: <b>0.04</b> Length: <b>.125</b>	Tolerance Bilateral Positive Tolerance Value: <b>.01</b> Negative Tolerance Value: <b>-.01</b> Precision: <b>3</b> Suppress lead Zeros
Arrowhead -Arrow2 Style: <b>Solid Arrow</b> Height: <b>0.04</b> Length: <b>.125</b>	Text Font: <b>iges1003</b> Height: <b>.125</b> Aspect Ratio: <b>1</b> Stroke Width: <b>.01</b> Orientations: <b>Horizontal</b> Display: <b>Centered on Leader</b>
Extension Line Line1: <b>Solid</b> Line2: <b>Solid</b> Gap Value: <b>.1</b>	
Leader Segment Leader 1: <b>Auto</b> Leader 2: <b>Auto</b>	Leader to Leader Minimum Clearance: <b>0.38</b> Leader to Feature Minimum Clearance: <b>0.5</b>
Extension Line/ Leader Segment Width: <b>0</b>	One Line Dual Units Format: <b>None</b>

Next, you start adding dimensions. You start by adding a cross-hair to the circle in the detail.

11. Choose **[Top Menu] Drawing > Add Cross-hair**. When prompted for a location, click on the circle in the detail drawing.

A cross-hair is added to the circle, and the prompt bar is removed. Next, you add a multi-vertex pointer to the circle, as shown in Figure 5-11.



**Figure 5-11. Pointer You Will Add**

12. Choose **[Top Menu] Drawing > Add Pointer:.** Enter **6** in the Text prompt of the prompt bar and TAB to the location prompt.
13. Place the cursor at the edge of the circle, exactly where you want the tip of the arrowhead, and click the Select mouse button.

A ghost image of a line displays, which follows cursor movements.

14. Place the cursor where you want the first vertex of the arrow leader and click the **Select** mouse button.

The first segment of the ghost image is fixed in place between the first two vertices.

15. Click where you want the end of the leader to connect to the triangle. Finally, **OK** the prompt bar.

The pointer is placed on the detail. Next, you change the style of the pointer you just added.

16. Choose **[Top Menu] Drawing > Change Pointer...** and click on the pointer. In the dialog box that displays, choose **Text Box Style Square** and press **OK**.

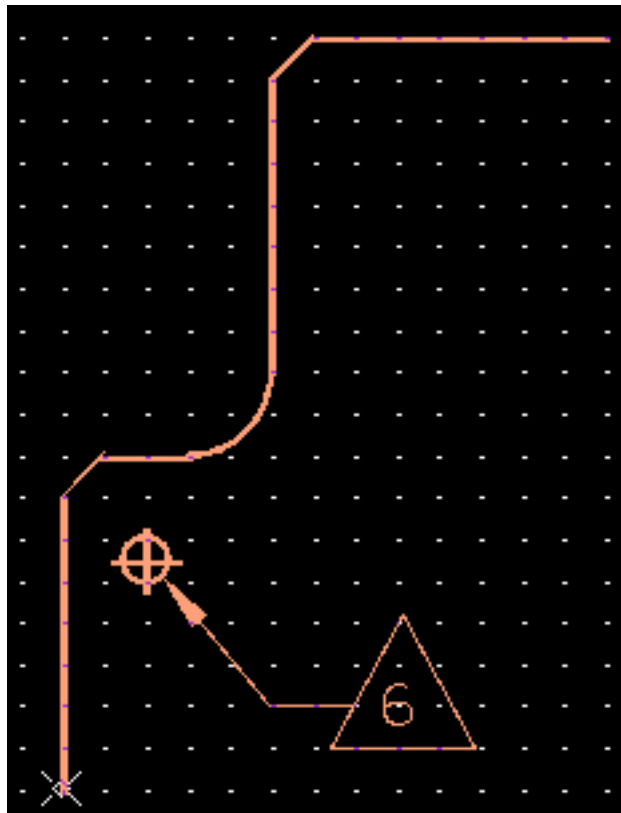
The pointer style is changed to a box, and the prompt bar is repeated so you can make additional changes.

17. Select the pointer again, and in the dialog box, choose the **Text Box Style Triangle**. Press **OK**, then **Cancel** the prompt bar.

The pointer style is returned to the triangle style.

18. Choose **[Top Menu] Drawing > Move**. When prompted to select something, click on the pointer. Move the cursor to reposition the pointer as you wish, and click the **Select** mouse button again to complete the move. **Cancel** the prompt bar.

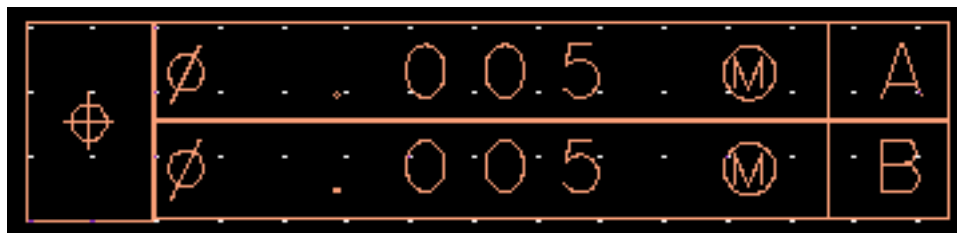
19. Change your pointer back to look like [Figure 5-12](#), if needed.



**Figure 5-12. Pointer Added to Detail**

20. Choose [Top Menu] Drawing > Add Feature Control Symbols. Study the dialog box that displays.

Create the Feature Control Frame shown in [Figure 5-13](#).



**Figure 5-13. Feature Control Frame**

This Feature Control Frame is made up of three separate boxes that you move together. You create the three boxes one at a time. The

first box you create is the box on the far left that contains only the circle with the centerline/cross-hair. The second box is the one at the top containing the text *.005* and *A* with other symbols. The third box contains the text *.005* and *B* with other symbols.

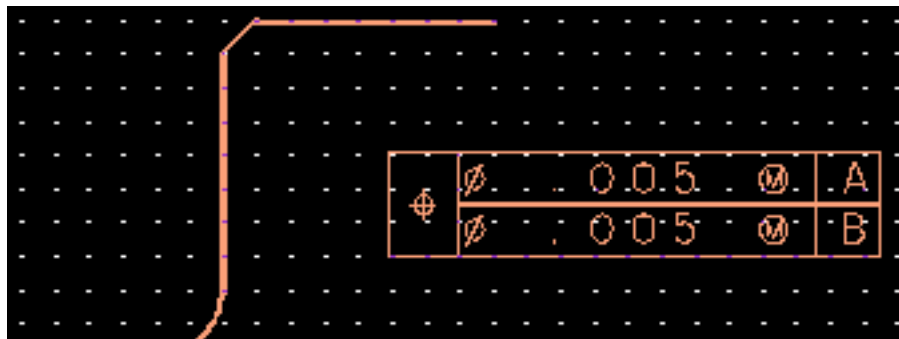
The dialog box shows the mapping of the feature control symbols to keyboard characters. You enter letters in the Text field to add the symbols you want. For example, if you want a circle with a centerline symbol, you enter *j* in the Text field.

If you want a vertical line to separate sections of a box, such as the vertical line between *.005* and *A* in [Figure 5-13](#), you enter a vertical bar: |.

21. Enter a **j** in the Text field of the Add Feature Control Symbols dialog box and press **OK**.

When you entered *j*, a circle with a centerline displayed in the Text field. Now you are prompted for a location, and if you move the cursor you can see a ghost image of the first feature control symbol box.

22. Move the cursor to place the ghost image in an empty area of the drawing where you want to place the feature control symbol, such as shown in [Figure 5-14](#). Click the Select mouse button.



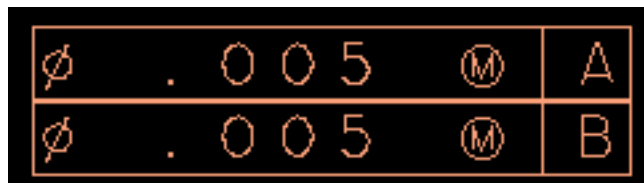
**Figure 5-14. Suggested Location for the Feature Control Frame**

The box with the circle and centerline is placed in the drawing and the dialog box is redisplayed for you to create another feature control symbol.

23. Remove the contents of the Text field (use the BackSpace key), and enter **n** followed by a space, followed by **.005**, space, **m | A**. **OK** the dialog box.
24. Place the image of this new feature control symbol to the right of the first one you created, without letting them touch.

You do not want the feature control symbols to touch yet, because you change the size of the first symbol later and then move them all together. The dialog box is repeated so you can enter the final feature control symbol.

25. In the Text field of the dialog box, backspace over the A to remove it, and enter **B**. **OK** the dialog box. Place this final feature control symbol directly under the last one, so they touch as shown in Figure 5-15. When the dialog box repeats, **Cancel** it.



**Figure 5-15. Two Feature Control Frames Placed Together**

Now you need to change the size of the first feature control symbol, with the circle and centerline, to match the height of the other two.

26. Choose **[Top Menu] Text > Change Text...** Click on the circle with centerline in the first feature control symbol. In the dialog box, enter the following values and press **OK**.

Box Style Height: **0.31**

Box Style width: **0.2**

The size of the box is changed, and the box is unselected. Next, you need to move the box.

As you saw, the Change Text dialog box includes an area to set the height and width of the box. Zeros are the default for the height and width, so the height and width are auto-scaled to fit the text.

27. Cancel the Select Area prompt bar that repeats.

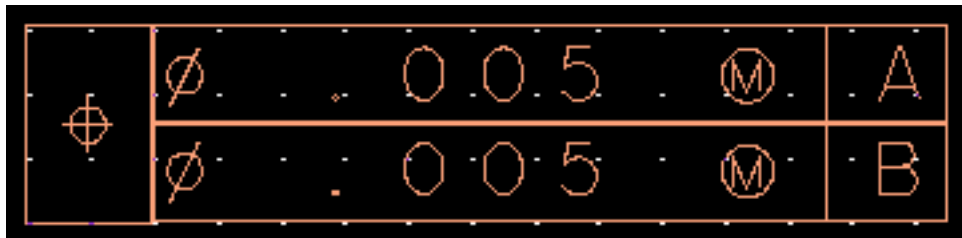
28. Choose **[Top Menu] Text > Move**. Click on the circle with centerline box and move the image to try to line it up with the end of the other two feature control symbols.

You probably cannot line the boxes up, because grid snapping is turned on and the box size does not match the grid. In the next step, while the box remains selected, you turn off grid snapping.

29. While the box remains selected, choose **Setup > Grid Snap Off**.

The grid snapping is now off, and since the box is still selected you can continue moving it.

30. Move the box again, this time lining it up perfectly with the edge of the other two feature control symbols, as shown in [Figure 5-16](#). Click the Select mouse button to place the box, then unselect all objects.



**Figure 5-16. Completed Feature Control Frame**

31. Choose **Setup > Grid Snap On**.

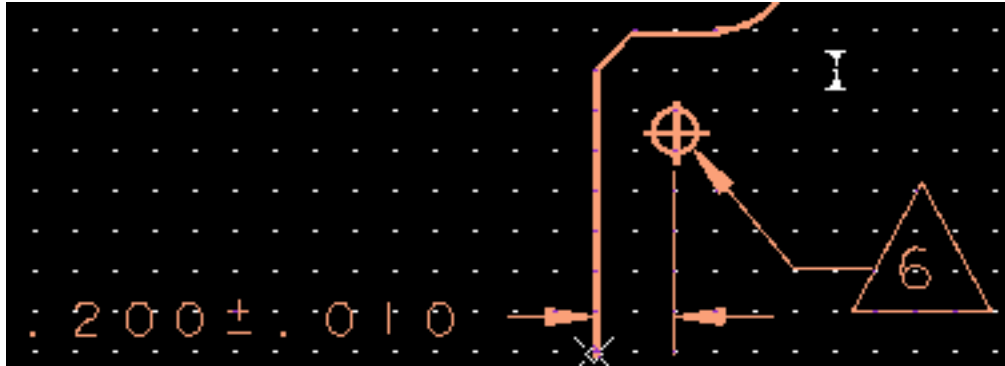
Next, you add a horizontal dimension.

32. Choose **[Top Menu] Dimension > Add Horizontal Dimension**. Click on the left vertical line near its vertex with the lower chamfer. Click on the center of the circle. Finally, place the ghost image of the dimension as shown in [Figure 5-17](#) and click again.

When you add horizontal, vertical, and angular dimensions, the default is for the dimensions to go to the vertices nearest to the locations you click on. When several vertices are close together, be



careful to click nearest the vertex to which you want the indicated dimension.



**Figure 5-17. Location of the Horizontal Dimension of the Circle**

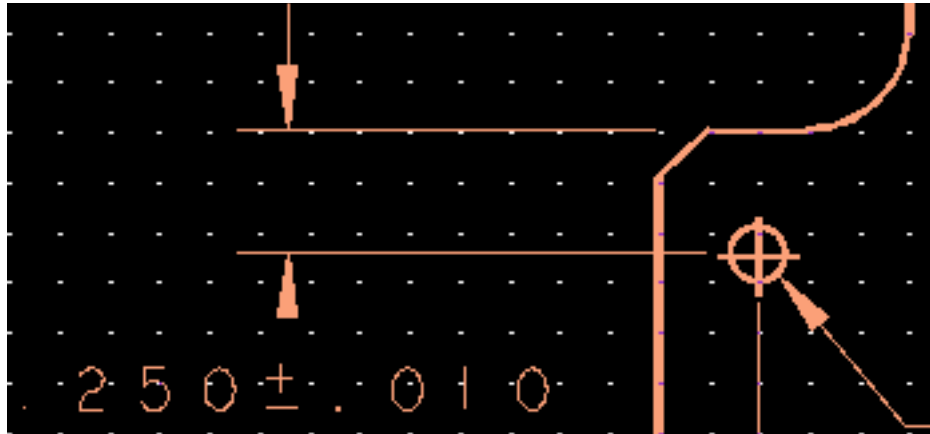
The dimension, extension lines, and arrows are added to the drawing. To add vertical dimensions, use the same process but choose **[Top Menu] Dimension > Add Vertical Dimension**.

33. Cancel the Add Horizontal Dimension prompt bar.

Next, you add the vertical dimension that specifies the location of the drill hole.

**34. Choose [Top Menu] Dimension > Add Vertical Dimension.**

Click on the horizontal line above the drill hole, near the chamfer. Next, click on the center of the drill hole. Position the image of the dimension as shown in [Figure 5-18](#).

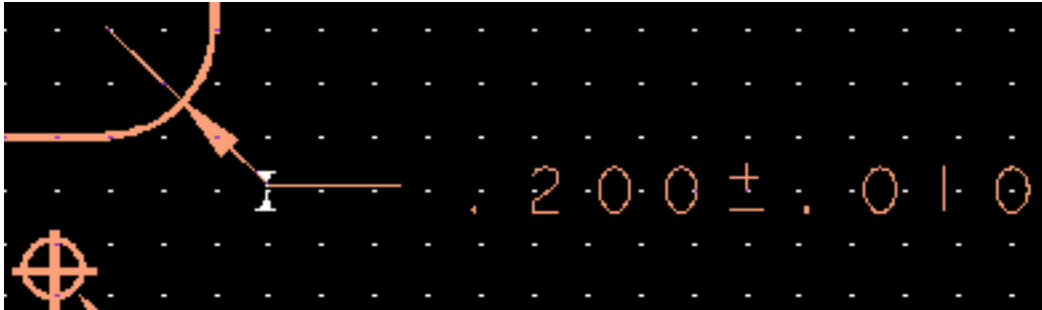


**Figure 5-18. Locating the Drill Hole Vertical Dimension**

**35. Cancel the Add Vertical Dimension prompt bar.**

Next, you add the dimension for the fillet.

36. Choose **[Top Menu] Dimension > Add Radial Dimension**. Click on the fillet. Next, position the ghost image so that the dimension is located as shown in [Figure 5-19](#) and click the Select mouse button again. Finally, cancel the Add Radial Dimension prompt bar.



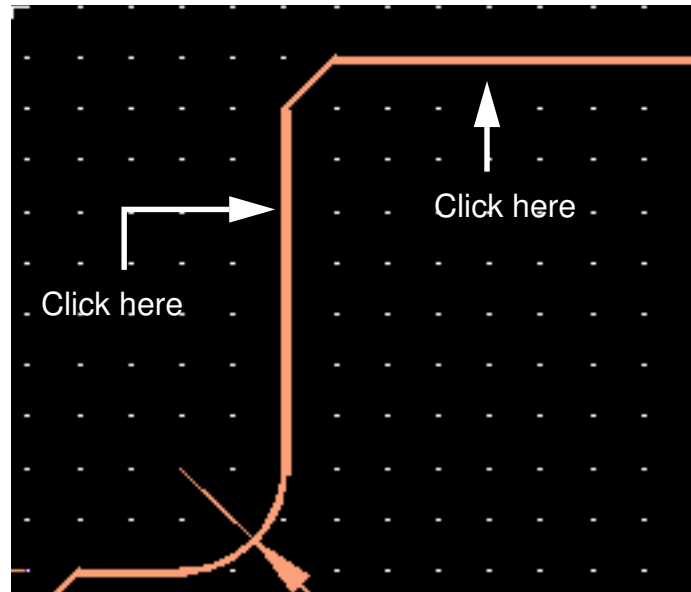
**Figure 5-19. Locating the Dimension of the Fillet**

The last dimension you add is for the chamfer. If you added the dimension for the chamfer using only the geometry currently available, you could show a 225 degree angle, or a 135 degree angle. What you probably want to show is a 45 degree angle. To show a 45 degree angle, you have to add a short horizontal extension line extending from the left end of the upper horizontal line of the detail. After adding the line you can add a dimension between it and the chamfer that shows a 45 degree angle.

Now, you add the horizontal extension line. You first change the line width to zero so that the line you add looks like the other extension lines.

37. Choose **Setup > Line Width**. In the prompt bar, enter **0** and press **OK**.

38. Choose [Top Menu] **Shapes > Add Line**. When the prompt bar displays, choose [Top Menu] **Shapes > Snap > Intersection**. When the prompt bar displays, click the Select mouse button on the two places shown in Figure 5-20.



**Figure 5-20. Locating Two Lines of an Intersection**

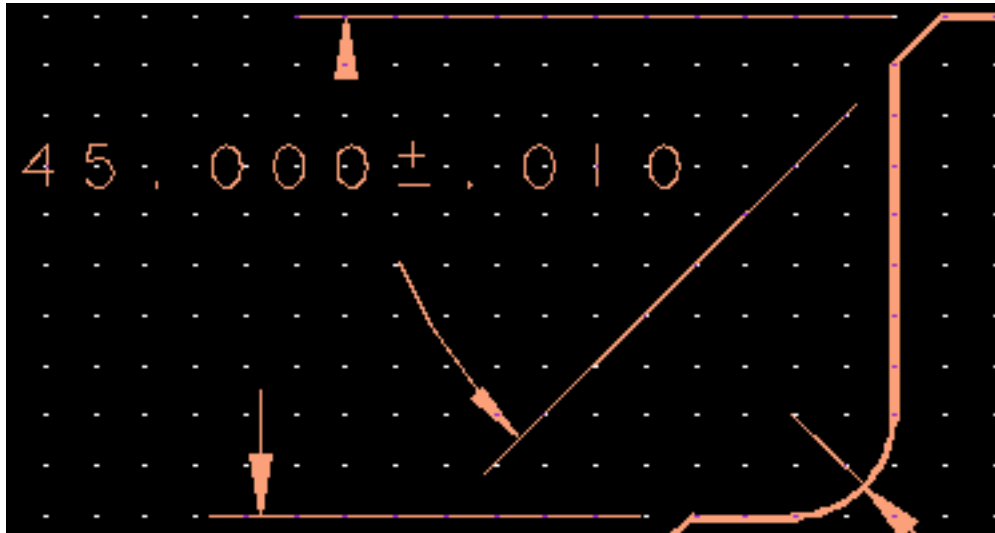
A basepoint is placed at the projected intersection of the two lines you clicked on. This is the first point of the line you are creating. Next, you specify that the other point of the line is -0.25 inches in the X direction from the basepoint.

39. Choose [Top Menu] **Shapes > Snap > Delta**. In the prompt bar, enter Delta X = **-0.25**, Delta Y = **0**, From **lastpoint**, and press **OK**.
40. When the Add Line prompt bar displays again, choose **OK** and if it repeats again, choose **Cancel**.

Now you are ready to add the angular dimension of the chamfer.

41. Change the line width setup back to 0.01.
42. Choose [Top Menu] **Dimension > Add Angular Dimension**. Click on the horizontal line you just added. Click on the angular line of the chamfer. Move the cursor to see the ghost image of the

dimension. Move the image between the horizontal line and the chamfer so the arrows are between the line and the chamfer, as shown in Figure 5-21. Click the Select mouse button. Cancel the prompt bar.



**Figure 5-21. Locating the Dimension of the Chamfer**

An angular dimension can show either the inside or outside angle depending on where you place the dimension text. In this example, the value of the inside angle (45 degrees) is shown. You could have shown the outside angle (315 degrees) by placing the text on the other side of the angle. Next you change the tolerance of the angular dimension.

43. Choose [Top Menu] **Dimension > Change Dimension**. Click on the angle dimension text. In the dialog box, enter the following values and press **OK**.

Precision: **1**

Positive Tolerance Value: **0.2**

Negative Tolerance Value: **-0.2**

The angle dimension is changed. Next you change the text setup and add some text to the detail to give it a title.

44. **Cancel** the repeated prompt bar.

45. Set up the text as follows:

Height: 0.15  
Font: std  
Box Style: Rectangle  
Justification: Center Center

46. Choose [Top Menu] Text > Add Text. In the prompt bar, enter **DETAIL A** and TAB to the location prompt. Move the cursor to position the ghost image of the text as shown in Figure 5-22. Click the Select mouse button. **Cancel** the repeated Add Text prompt bar.

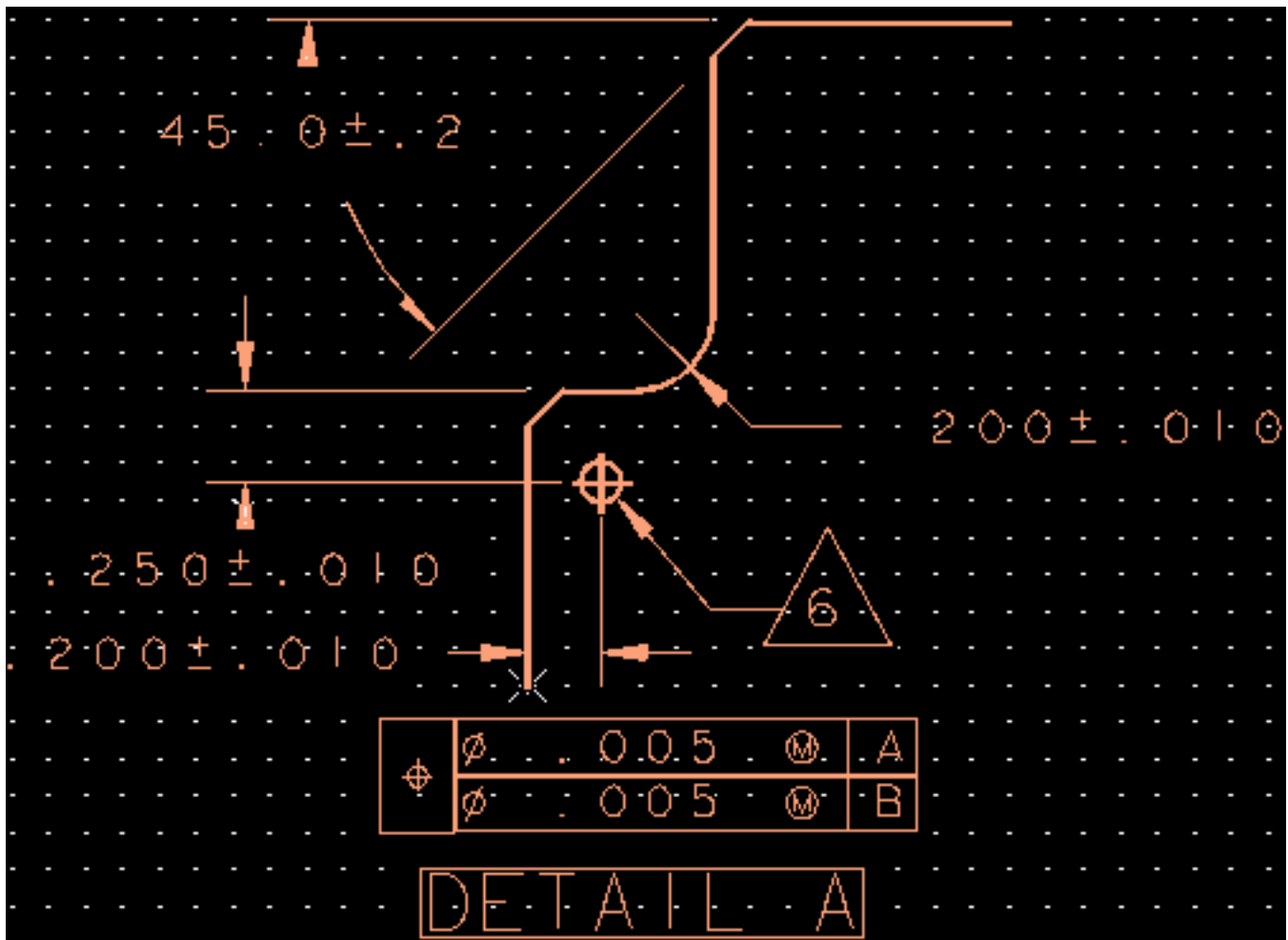


Figure 5-22. Completed Detail

## Creating a Fabrication Drawing

You are ready to add the detail drawing to a fabrication drawing for your design. In this section, you create a complete fabrication drawing. FabLink sets up the drawing by getting the format sheet outline from a standard library. You need to add the required text to the drawing format. You also need to add the board image and your detail drawing to the drawing format. To provide more dimensioning practice, you dimension the board outline for manufacturing. You add notes and a drill hole schedule to complete the drawing. At the end of the lab, [Figure 5-29](#) and [Figure 5-30](#) show an example of the fabrication drawing.

Some of the drawing geometries required for the fabrication drawing, such as the title block and sheet border, are provided in your training data. FabLink follows the variable pathname `$MGC_PCBPARTS`, defined in your `location_map` file, to locate these geometries. You must change the value of the `$MGC_PCBPARTS` variable pathname to equal the pathname to the training data so that the geometries in the training data are used by FabLink. The change you make to the `$MGC_PCBPARTS` variable is temporary and works only in the current FabLink session. You do not change the permanent `location_map` file.

A `location_map` is an ASCII file containing variable pathnames, such as `$MGC_PCBPARTS`, and their *hard* pathname values. These provide access to parts libraries, the MGC tree that contains the MGC software, and so on.

1. Choose **MGC > Location Map > Change Entry**. In the Soft Name field of the prompt bar, enter: `$MGC_PCBPARTS`. TAB to the Hard Name field, and enter: `your_path/training/board_new/mod7/sig_az/pcb_parts`. Press RETURN, or **OK** the prompt bar.

The variable pathname `$MGC_PCBPARTS` uses the `your_path/training/board_new/mod7/sig_az/pcb_parts` hard pathname. You verify this in the next step.

2. Choose **MGC > Location Map > Show Location Map**.

A Location Map window displays.

3. Activate the Location Map window, and scroll to see the \$MGC\_PCBPARTS entry and its Hard Name value.

\$MGC\_PCBPARTS must be set to your pcb\_parts directory in your training directory. If the word *overridden* is next to the Hard Name, it means the \$MGC\_PCBPARTS soft pathname has been temporarily changed from its preset value in your location\_map file. It resets to the value defined in your location\_map when you close the FabLink session.

4. Close the location map window

Now you are ready to create the fabrication drawing.

5. Choose **Geometries > Create Geometry > Drawing...**, enter the following values in the dialog box, and press **OK**.

Mentor Formats  
Drawing Name: **fab\_dwg**  
Size: **C**  
Create Sheet: **Sheet 1**  
Add Approval Block  
Add Revision Block

A new edit window, named DR\$fab\_dwg, displays. A report window might also display.

6. Close the report window, if one displays.
7. Activate the window DR\$fab\_dwg, then set up the view layers to make the Board\_outline, Drawing\_1, and Drill\_holes layers visible.

Most of your work is on the Board\_outline, Drawing\_1, and Drill\_holes layers.

8. View the area around the title block.

You add a title in the title block.

9. Set up the text so Box Style equals none, and font is Leroy. Leave all other text styles as they are.



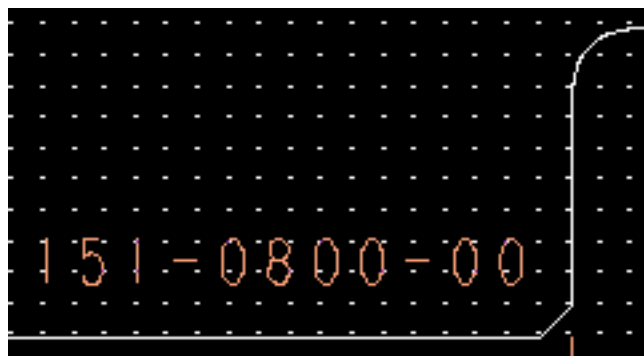
10. Choose **[Top Menu] Text > Add Text**. In the prompt bar, enter **PWB FABRICATION DRAWING** in the Value prompt, and TAB to the location prompt. Place the text in the upper half of the TITLE field of the title block.
11. Using the prompt bar that repeats, add the text: **SIGNAL ANALYZER** in the lower half of the TITLE field of the title block. Add *1 OF 1* to the sheet block and the letter *A* to the revision block. Add your name to the box above the title.

You can add any other appropriate text you want. To experiment, you can change the size and style of any added text. Next, you add the board image to the drawing.

12. View all of the drawing.
13. Choose the **[Top Menu] Drawing > Add Board**. Move the cursor in the edit window to see the ghost image of the board. Place the board image in the right half of the drawing and click the Select mouse button. **Cancel** the repeated prompt bar.

You do not have to provide the board pathname, because you already have the board geometry open in this session of FabLink.

14. Add a part number of: 151-0800-00. Place it in the lower-right corner of the board outline, as shown in [Figure 5-23](#).



**Figure 5-23. Part Number Text Added To Board Outline**

15. Cancel the Add Text prompt bar when it repeats.

16. Add the vertical dimensions shown in Figure 5-24.

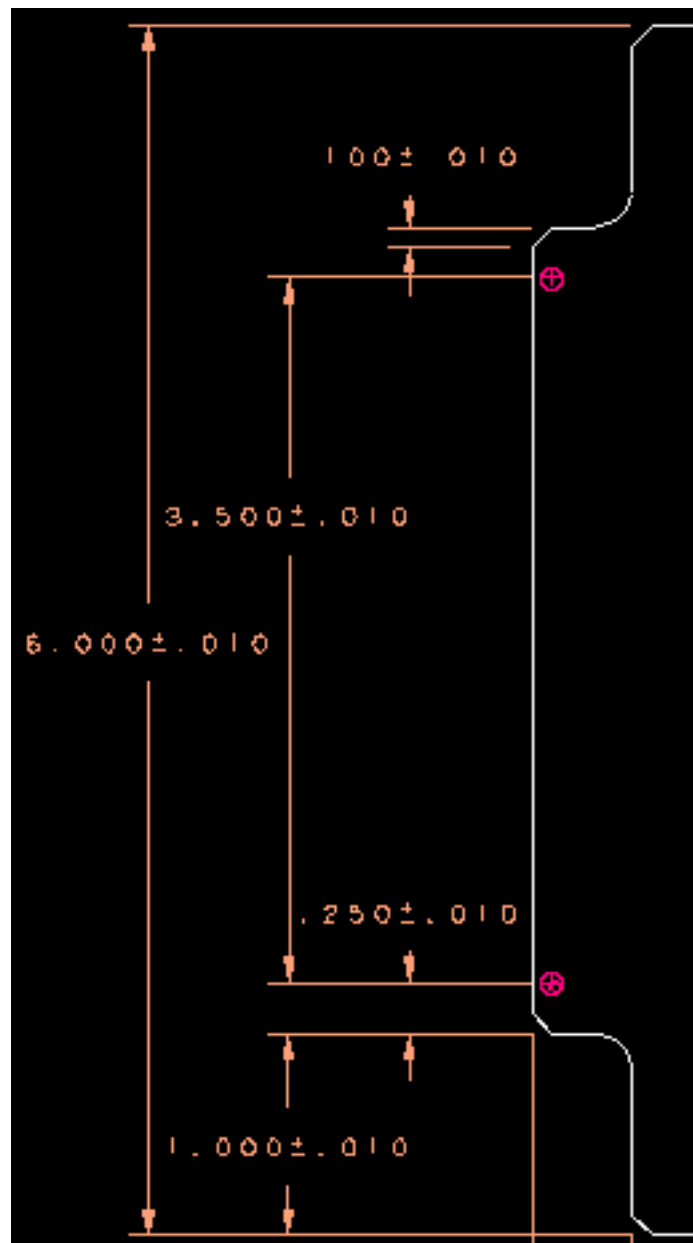
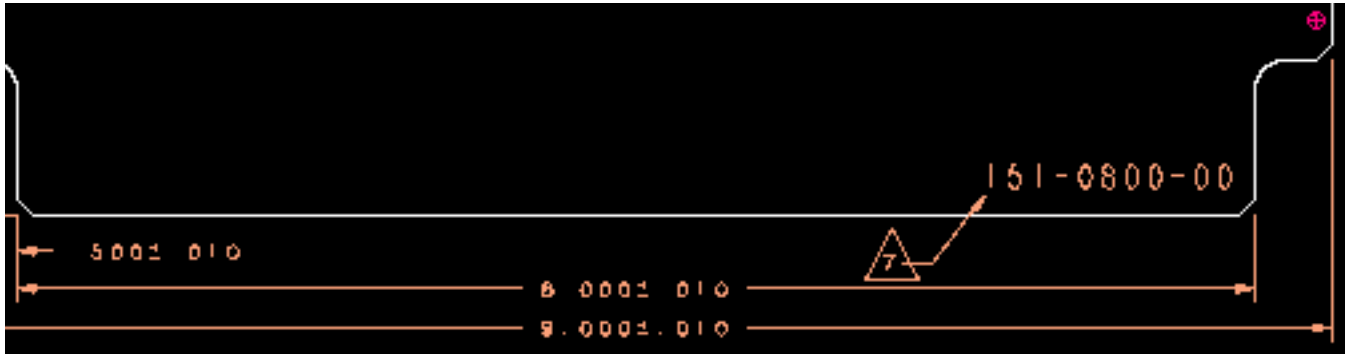


Figure 5-24. Board Outline With Vertical Dimensions

17. Add the horizontal dimensions to the board, and add a pointer to the board's part number. Use a triangular box and the number 7 as text. Refer to [Figure 5-25](#)

The numbers in the figure might be difficult to read, but the values are not important. When you add the dimensions, you see the correct values.



**Figure 5-25. Board with Horizontal Dimensions and Pointer**

If you have time, you can experiment with dual dimensions, leading and trailing text placement, and other dimension placement features. You can set up these features using **Setup > Dimension**.

18. Choose **[Top Menu] Shapes > Extended Menu > Add Geometry**. In the prompt bar Geometry field, enter: **detail\_a**. TAB to the location prompt. Move the cursor in the edit window, and place the detail\_a image in the lower-left quadrant of the drawing. **Cancel** the prompt bar that repeats.
19. Set up the text so the justification is Center Left.

You are going to add a block of text from a file. Each line of the text is justified according to the setup.

20. Choose **[Top Menu] Text > Add Text File:.** In the prompt bar, enter the pathname: `your_path/training/board_new/mod7/fab_notes`. TAB to the location prompt. Move the cursor into the edit window, then position the image of the text in the upper-left quadrant of the drawing and click the Select mouse button.

21. Set up the line width to 0.

22. View a very small area around the numbers 6 and 7 in the list of notes you just included.

You need to draw triangles around the numbers 6 and 7 in the notes, because the numbers refer to pointers on the drawing.

23. Choose **[Top Menu] Shapes > Add Line.** Add a line to form a triangle around the number 6 in the list of notes. Add another line to form a triangle around the number 7. Refer to [Figure 5-26](#). **Cancel** Add Line prompt bar.



**Figure 5-26. Triangles Added Around Numbers in List**

If you used **[Top menu] Shapes > Add Polygon** to create the triangles, the triangles are filled, and you must change the view style to make polygons unfilled, which might change something else on the drawing. It is easier to just add a line to form a triangle.

In the next few steps, you prepare, create, and add a drill schedule to the drawing.

24. Choose the **Pop** menu item in the Window Menu (the icon in the upper-left corner of each edit window) to pop the edit window and reveal another edit window. Sometimes you have to pop the window twice. Continue popping edit windows until the board geometry is visible.

If you saved the milling information, only the milling layer might be visible in the board geometry. The **Drill Menu** is associated with the board geometry. Next, you restore the changes you made to the drill table earlier.

25. Choose **File > Restore > From Design...**, select **Drill Table** in the Restore From Design dialog box, and press **OK**.

If you see a message stating that this functionality is not available without the Testpoints option, ignore it. The message refers to a different drill table. Next, you specify geometries as drill symbols in the drill schedule.

26. Choose **[Top Menu] Drill > Change Drill Table > Change Drill Table...**

The Change Drill Table dialog box displays. Notice that the Symbol column is empty. In the next steps, you read in drill symbols and assign the symbols to drill positions. Because you have not yet read in the drill symbols, you must do that before you can assign values to the Symbol column.

27. Close the Change Drill Table dialog box.

28. View the contents of the mgc.trng.drawings User library. Select the first four drill geometries (drill1 through drill4) and read them.

Four edit windows display as the drill geometries are read.

29. Close the four drill geometry edit windows.

Now that the drill geometries are read into the session, you can change the drill table.

30. In the Board geometry edit window, choose **[Top Menu] Drill > Change Drill Table > Change Drill Table...**, select Position 1, and click on **Change**.

In the Change Drill Information dialog box, enter **drill1** for the Symbol Name, **200** for the Feed Rate, and **300** for the Speed Rate, then press **OK**. Verify that the symbol information is changed by reading the Symbol column. Add a drill symbol, feed rate, and speed rate (specify any drill symbol from drill1 through drill4) to each of the other drill positions. When finished, **Close** the dialog box.

Drill symbols are generic geometries, so you can create your own drill symbols if you want. Be sure that your assigned symbol names appear correctly in the right-hand column of the Drill Table. If your design choices vary slightly from this example, you might not need all four drill symbols.

31. Pop the edit windows again to return to the DR\$fab\_dwg edit window containing the fab\_dwg geometry.

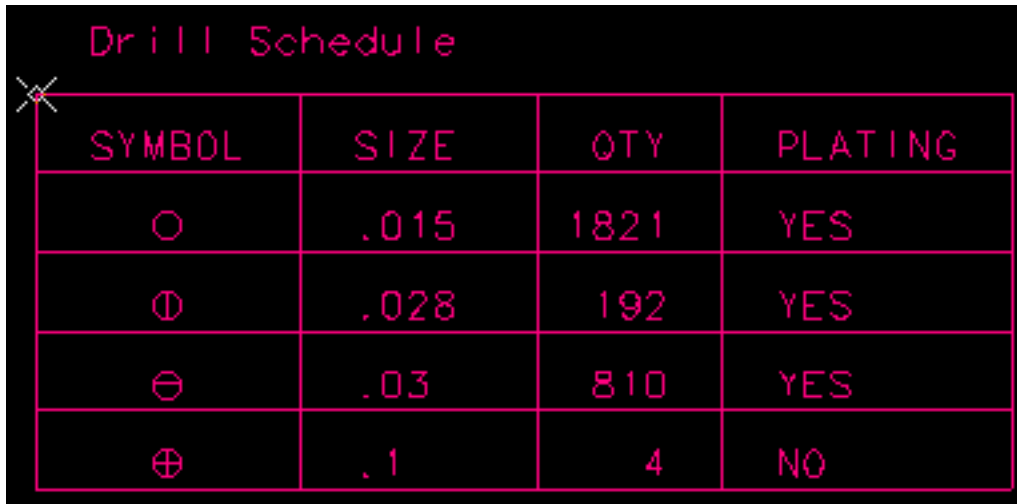
Next, you create a drill schedule.

32. Choose **Geometries > Create Geometry > Drill Schedule**, enter the following values in the dialog box, and press **OK**.

Create Drill For: **Board**  
Title for Board Drill Schedule: **Drill Schedule**  
**Master Schedule**  
Drill Symbol Position: **1** Title: **SYMBOL**  
Drill Size Position: **2** Title: **SIZE**  
Count Position: **3** Title: **QTY**  
Plated Position: **4** Title: **PLATING**

Enter **0** (zero) for all other positions so no other columns are made. Leave all other entries default.

A new edit window named DR\$drill\_schedule\_master displays, containing a drill table that looks similar to the one in [Figure 5-27](#).



SYMBOL	SIZE	QTY	PLATING
○	.015	1821	YES
⊖	.028	192	YES
⊖	.03	810	YES
⊕	.1	4	NO

**Figure 5-27. Customized Drill Schedule**

**33.** Close the drill schedule edit window.

**34.** View all of the drawing in the fab\_dwg edit window.

Next, you add the drill table to the drawing.

**35.** Choose **View > Drill Symbols On** to make the drill symbols visible on the drawing.

**36.** Choose **[Top Menu] Drawing > Add Drill Schedule:.** In the Add Drill Schedule dialog box, select the drill schedule you just created (drill\_schedule\_master), and press **OK**. When prompted for a location, move the cursor in the edit window to see the size of the image.

To make the drill schedule larger, Tab to the Scale prompt and change the value. A good size is 1.0. Tab again to the location prompt, move the image of the drill schedule to a position anywhere in the drawing, and click the Select mouse button.

Congratulations! You just completed a fabrication drawing using FabLink.

## Creating an Assembly Drawing

In this section, you create a complete assembly drawing for your board. After using FabLink to set up the drawing and the format sheet outline, you add the required text and the board image to the drawing format. You complete the drawing by adding a side view of the board and a customized bill of materials. An example of the completed assembly drawing is included at the end of this section in [Figure 5-31](#) and [Figure 5-32](#).

1. Create a new drawing named *assy\_dwg*. It must be a C-size drawing sheet with a parts list, revision block, and approval block. If you need help, refer to step 5 of section "[Creating a Fabrication Drawing](#)" on page 5-37.
2. Close the report window (if displayed). Activate the new edit window. Set up the text to have no box style.
3. Add the text for the title block. Use the title: ASSEMBLY DRAWING, SIGNAL ANALYZER. Add the sheet number, revision, and your name.
4. Cancel the Add Text prompt bar when finished adding text.
5. Make the PLACE\_1, BOARD\_OUTLINE, and DRAWING\_1 layers visible.
6. Add the board geometry in the lower-left corner. Leave some space to the right of the board geometry so you can later include a right-side view beside the board. Do not add a reference designator. Refer to step 13 of section "[Creating a Fabrication Drawing](#)" on page 5-37 for help in adding board geometry. After you finish, cancel the prompt bar that repeats.

You do not need to add a reference designator, because these were added to the PLACE\_1 layer.

7. Add the part number of 151-0801-00 to the lower-right corner of the board, as you did in the fabrication drawing. Cancel the Add Text prompt bar when it repeats.



8. View the area around the right-side connector, leaving enough room in the view to add a pointer in the next step.
9. Add a pointer to the right-side connector. Use a 2 enclosed in a triangle for the text.
10. View all of the drawing.
11. Add the assembly drawing notes from the file: your\_path/**training/board\_new/mod7/assy\_notes**. Place the notes in the upper-right corner of the drawing.
12. Draw a triangle around the number 2 in the list of assembly notes, because this is a reference to the pointer.

In the next few steps, you add a side view of the board to your assembly drawing. FabLink can automatically create this view for you, but before it does, you set the line width.

13. Verify that the line width is set to zero (0).

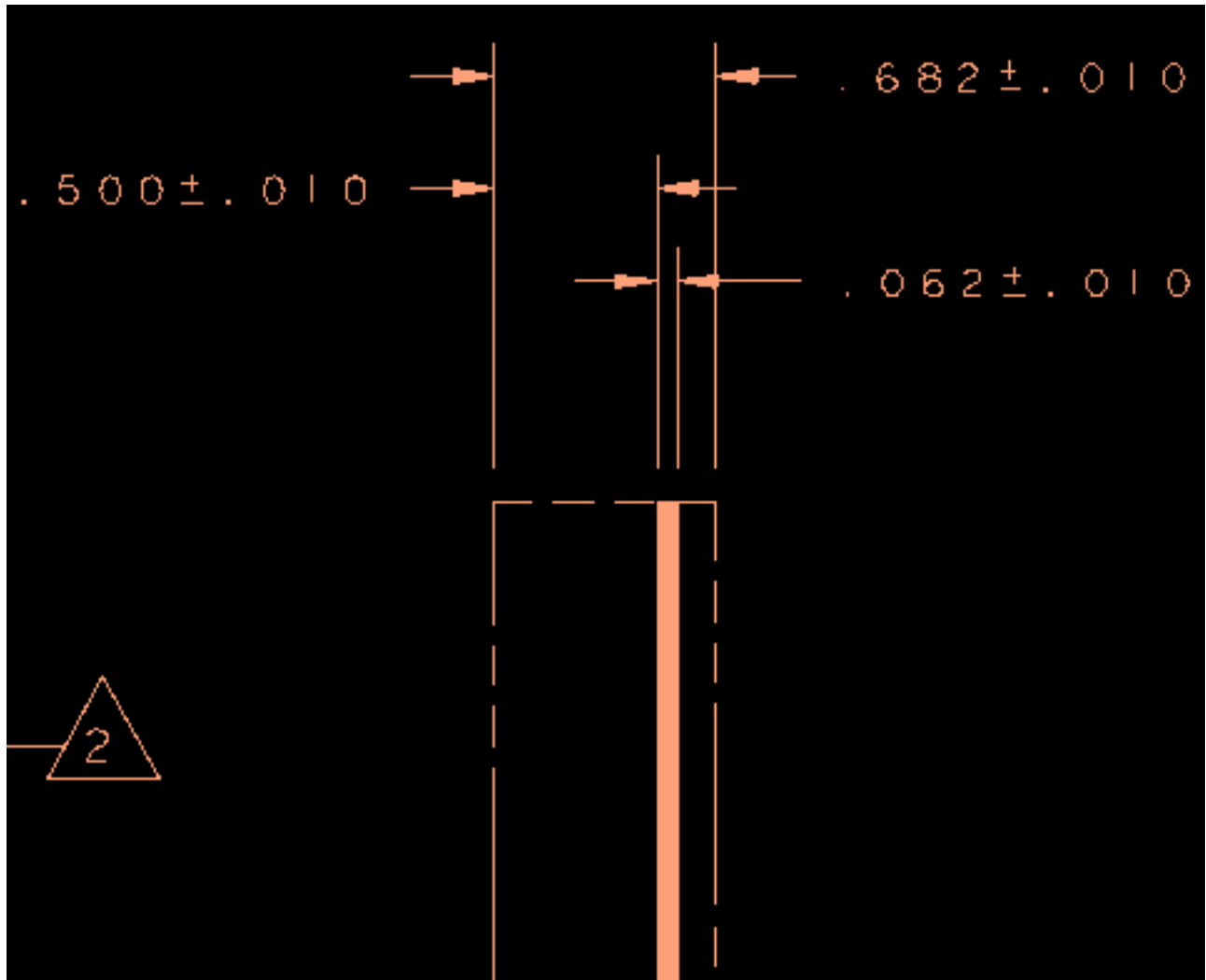
Line styles, like the request for a phantom line in the next step, are available only for zero width lines.

14. View all of the drawing.
15. Choose **[Top Menu] Drawing > Add Board Side View:**. In the prompt bar, click on **Options**. In the Add Board Side View (Options) dialog box, enter the following values and press **OK**.

Board Thickness: **.062**  
**Include Component Height**  
**Specify Component Height**  
Component Height Front: **.5** Back: **.12**  
Overall Thickness Line Style: **Phantom**

16. Move the cursor into the edit window. Place the ghost image of the side view of the board to the right of the board. Click the Select mouse button.

17. Complete the side view of the board by adding the horizontal dimensions shown in Figure 5-28.



**Figure 5-28. Board Side View**

Next, you create a bill of materials.

18. Choose **Report > Bill of Materials...**

In the next few steps, you create a customized bill of materials and add it to the board as text from a file.

19. Enter the information from [Table 5-7](#) into the Report Bill of Materials dialog box. Leave all other fields in the dialog box set to default values and press **OK**.

**Table 5-7. Report Bill of Materials Dialog Box Entries**

To Pathname: your_path/training/board_new/mod7/bill_of_materials		
<b>Part Number</b>		
	Position:	Title:
Item Number	6	ITEM NO.
Part Name	2	PART NUMBER
Geometry	5	GEOMETRY NAME
Count	1	QUANTITY
Description	3	DESCRIPTION
Reference Name	4	REF DES
Comment	0	COMMENT

A note in the message window states that the bill of materials data was written. Next, you include that file in the drawing.

20. Set up the text to use a height of 0.05 inches. Use the leroy font.

Some of the characters in the bill of materials become special symbols with the iges1003 font, which is hard to read, so use the leroy font. The std font is also legible.

Later, if you do not like the size or aspect ratio of the bill of materials, you can delete it from the drawing, change the setup of the text size, and then re-include the bill of materials.

- 21.** Include the bill of materials text file, using the pathname used when you created the bill of materials. Use the method you used to add the assy\_notes text file. Place the bill of materials wherever it fits in the drawing. To change the size, change the font size before you place the text image in the drawing.

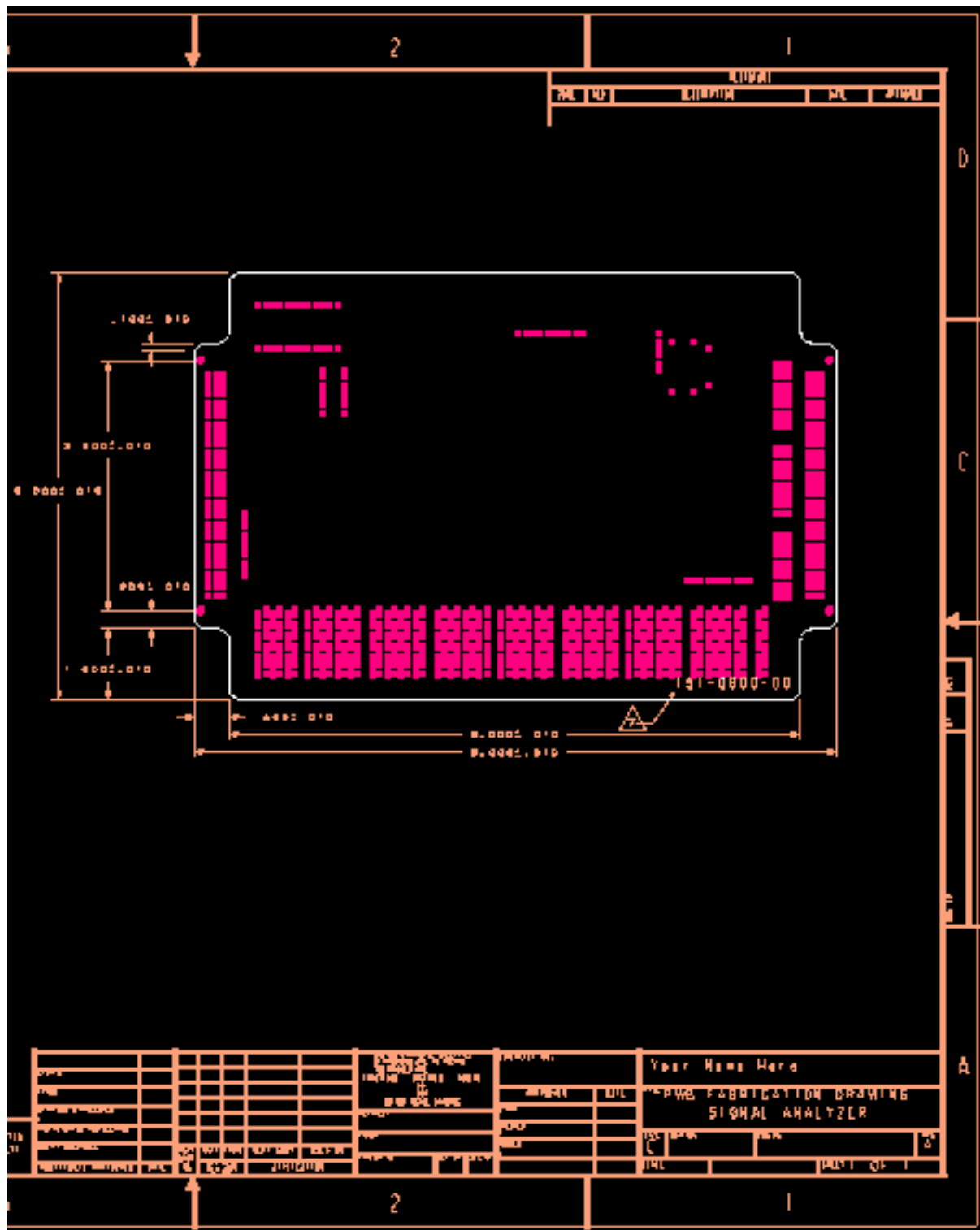
Your drawing is complete. If your company creates fabrication and assembly drawings differently from what you have just created, you can modify your lab exercise drawings.

You can also check with your instructor or system administrator about plotting your completed drawings.

- 22.** Save your design and close the FabLink session.

You have completed the "Creating Fabrication and Assembly Drawings" lab exercise. Continue with Lesson 6, "Releasing a PCB Design".





### Figure 5-30. Completed Fabrication Drawing

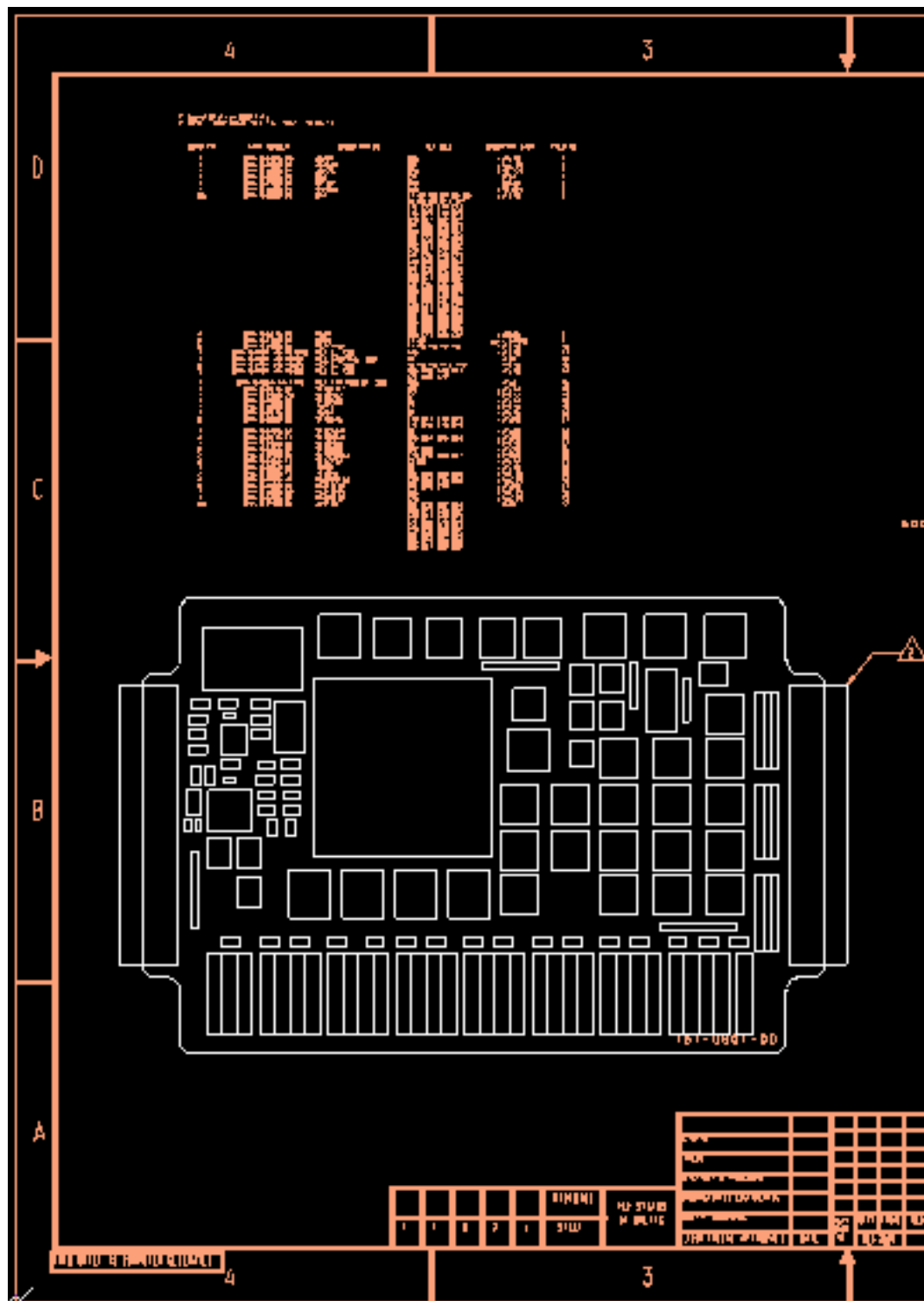
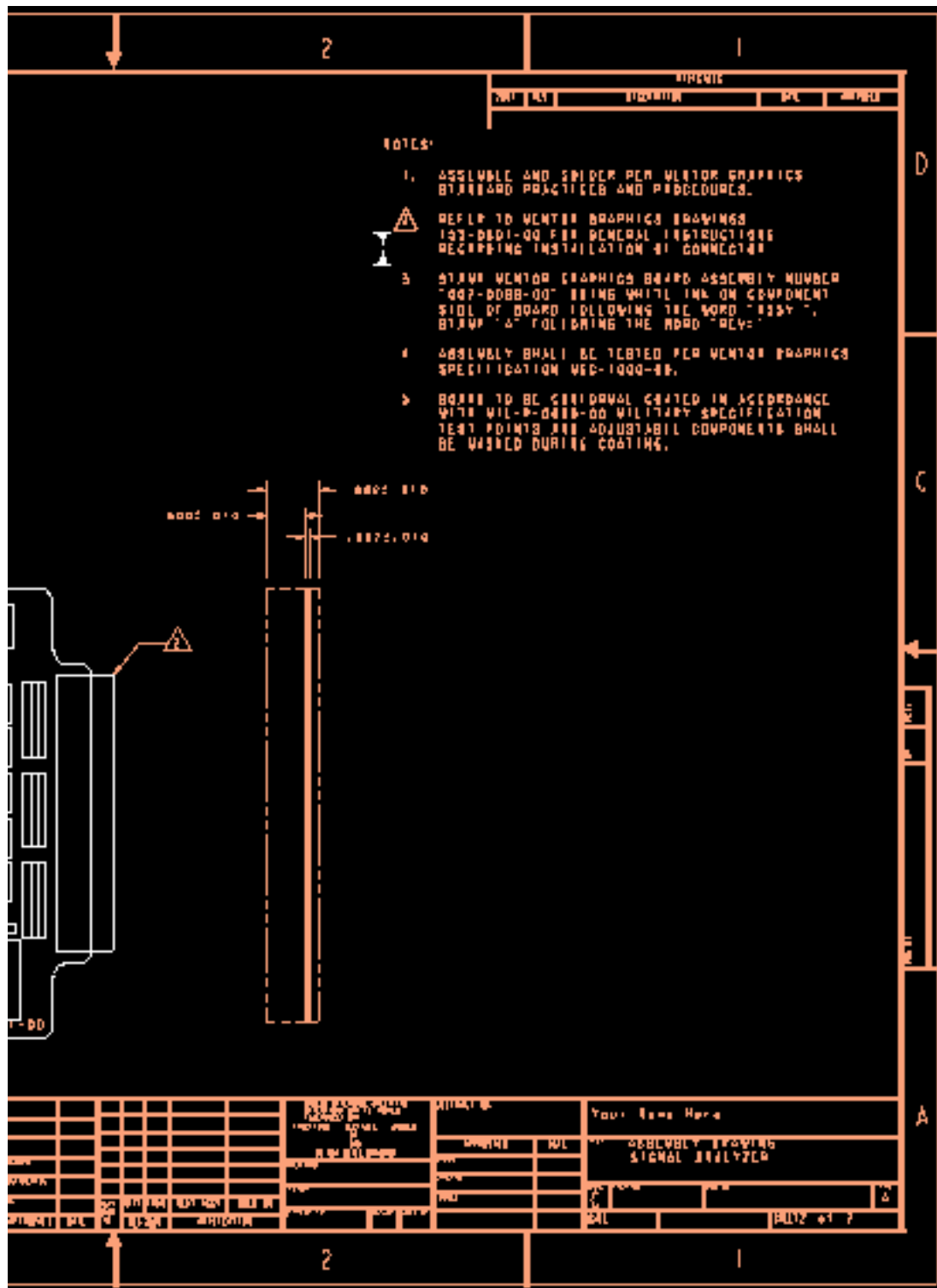


Figure 5-31. Completed Assembly Drawing



### Figure 5-32. Completed Assembly Drawing



# Lesson 6

## Releasing a PCB Design

This lesson discusses how to configure the elements of a design for releasing, archiving, and other operations.

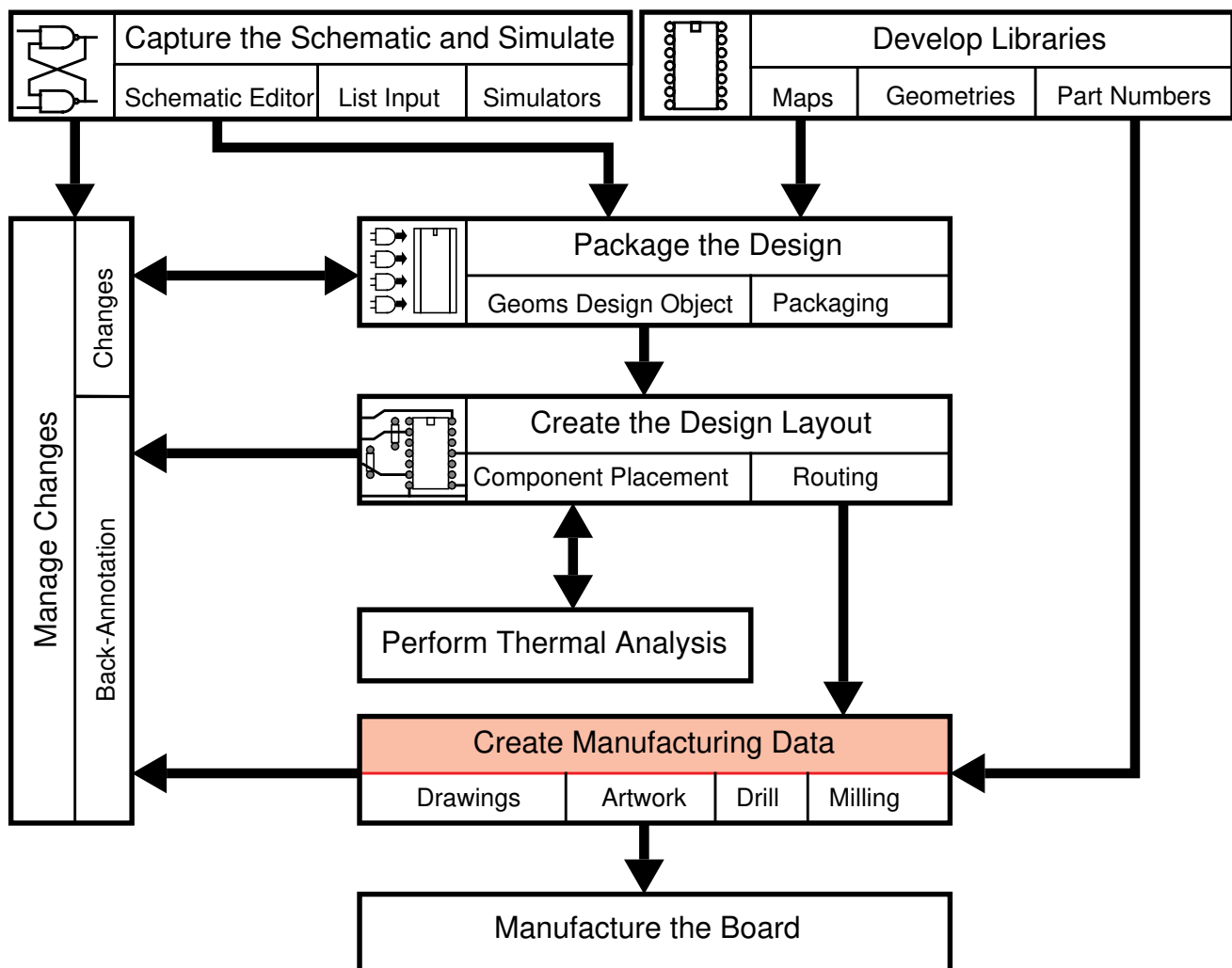


Figure 6-1. Board Process Flow Chart

## Objectives

In this lesson, you learn how to use Design Manager to gather a set of elements into a design configuration, and how to release that design configuration to a protected area.

After completing this lesson and lab exercise, you are able to:

- Describe the two methods of releasing PCB designs.
- Describe what a design configuration is, and is not.
- Create a customized design configuration.
- Release a design configuration.
- View a design configuration in different ways.
- Freeze a design object or a design configuration.

## Release Methods

Design Manager provides two methods of releasing PCB designs. The first is *Release PCB*, which is an automated utility that lets you release or transfer self-contained PCB-specific design data. Release PCB is discussed in the next subsection.

The second method involves building and releasing a protected, self-contained copy of a design and its contents, called a *design configuration*. This method, which you use in the lab exercise, is described in detail beginning with "[Design Configuration Concepts](#)" on page 6-8.

## Release PCB

You can release PCB data automatically, using a defined directory structure that ensures the inclusion of PCB-specific data, by selecting **Release > PCB** from the menu bar in Design Manager and completing the dialog box that appears. The Release PCB Design dialog box lets you copy or release PCB design data into three standard formats, which automatically collect and configure the data for you.

The Release PCB standard formats release an entire design's data, only PCB data, or only the schematic.

## Releasing the Entire Design

This format creates a comprehensive release of your design that includes the following data:

- *pcb* container and its dependent files.
- *schematic* container.
- schematic models for each instance referenced.
- a choice of three levels of design-specified catalogs and associated mapping files—all catalogs and mapping files, those valid for the design, and only those used in the design.
- *design\_geom* directory, which might include design-specific ASCII geometry data.
- *design\_lib* directory.
- *local\_lib* directory if directed via the location map.
- viewpoints.
- *startup* directory.

Release PCB releases schematic symbols in the same working directory as the design with these exceptions:

- If the -type LIBRARY keyword is used in the location map of a used library, then symbols from that library used in the design are released into a *local\_lib* directory under the design directory.
- If symbols are contained in the *design\_lib* directory, then they remain in the same directory of the released design.

Because this format is comprehensive, creating a stand-alone and platform-independent design archive, use this option when you want to perform these tasks:

- ***Archive a design.*** Releasing the entire design lets you later restore the design completely (for example, when you need to retrieve data to respond to engineering changes or defect reports).
- ***Create a design snapshot.*** Releasing the entire design lets you maintain a record of the design for use with checkpoints or milestones that you might have in your design cycle.
- ***Reuse a design for a new project.*** Releasing an entire design is useful when you want to start a new project using existing design data.
- ***Transfer a design.*** Because the design is platform-independent, you can copy the design to an internal or external design group that uses Mentor Graphics PCB tools. Sending the entire design allows updates to the schematic through back annotation. You can then import the back-annotated data to the original design directory using DVE.

## Releasing Only PCB Data

This format releases only the following PCB-related data:

- *pcb* container and its dependent files.
- *startup* directory.
- a choice of three levels of design-specified catalogs and associated mapping files—all catalogs and mapping files, those valid for the design, and only those used in the design.
- *design\_geom* directory, which might include design-specific ASCII geometry data.

This format does not include the PCB design viewpoint (*pcb\_design\_vpt*) and back annotation object, but does allow the transfer of design data in the form of a netlist. Use this format when you want to perform these tasks:

- **Transfer PCB data.** Transferring PCB data can provide information to an internal PCB fabrication or assembly group, to a service bureau, or to an external layout group.

You can transfer PCB data to an internal PCB fabrication or assembly group to drive the physical layout process. You can also transfer PCB data to a PCB service bureau or external layout group that does not use Mentor Graphics layout tools. Releasing only the *pcb* container allows the service bureau or external layout group to place and route the board but not to change the schematic, thus ensuring the integrity of the schematic data.

- **Archive only PCB design data.** This is used if you do not want to copy the entire data structure of your design, such as the *schematic* container and the schematic models for each referenced instance. You can copy only the design data related to the physical layout and manufacturing of your design.

## Releasing Only the Schematic

This format releases only the *schematic* container and all its associated models, and the format excludes all corresponding PCB data. Use this format to perform these tasks:

- ***Transfer the schematic.*** Transferring the schematic to other groups within your company continues the design process. It can include transferring to another group for simulation, design reuse, or layout.
- ***Copy the schematic.*** Copying the schematic enables design reuse or design experimentation.
- ***Archive the schematic.*** Archiving the schematic, with or without symbols, stores the logic of your design for reference later.

## Release PCB Summary

The Release PCB formats release current versions of design data. *Layering* is the process of copying in subsequent releases additional versions of design data to the same destination directory, releasing new versions on top of the existing versions. Layering the *schematic* container is supported, while layering geometry and mapping files is not. Catalog and mapping files use symbolic links for versioning, which Design Manager, in turn, views as files rather than versioned objects.



*While layering catalog and mapping files, you can overwrite design data.*

Releasing designs using Release PCB creates a single version of each design object. The first time you release design objects to the destination directory, Design Manager assigns a version of 1 to all versioned design objects. In subsequent releases to the same destination directory, Design Manager layers new versions on top of the existing versions, creating a version history of the released configuration.

You can also include in your release design-pertinent information that is not located in the directory hierarchy of the design you are releasing. This option allows you to reference and release additional data.

Additional data can be technology files or documentation associated with the design. Conversely, you can prohibit design object data from being released with the design data.

Figure 6-2 shows default release structures for each Release PCB format.

Release Entire PCB	Release Only PCB	Release Only Schematic
<div>design   schematic   pcb   pcb_design_vpt   design_maps     design catalog     symbol.map   local_maps   design_geom   local_lib     part1     part2     .     .     .   configuration   location_map   startup   additional data*</div>	<div>design   pcb   design_maps     catalog_files     symbol.map   local_maps   design_geom   configuration   location_map   startup   additional data*</div>	<div>design   schematic   local_lib     part1     part2     .     .     .   configuration   location_map   additional data*</div>

additional data\*—refers to data not located in the directory hierarchy of the design to release.

Figure 6-2. Default Release Structures for Releasing PCB Data

For more information about using Release PCB, refer to the *Design Manager User's Manual*.

## Design Configuration Concepts

Before you can release a design, you must create a configuration that includes the design data and all related elements. In Design Manager, the term *configuration* refers to a set of design objects that are related by containment or references.

Most PCB design data is located in a single container; however, some data is referenced. For example, component symbols on your schematic are usually included by reference, and are located on another workstation. Design object references allow you to distribute your design data across many locations in the file system. This feature can also make operations on the entire design as a single unit very difficult. To solve this problem, Design Manager provides *design data configuration management*, which consists of two items:

1. **Configuration window.** This window, shown in Figure 6-3, is a graphical tool you use to gather all the elements of a distributed design into one place. After you create a configuration, you can perform several different operations on the entire design, such as copying and releasing.

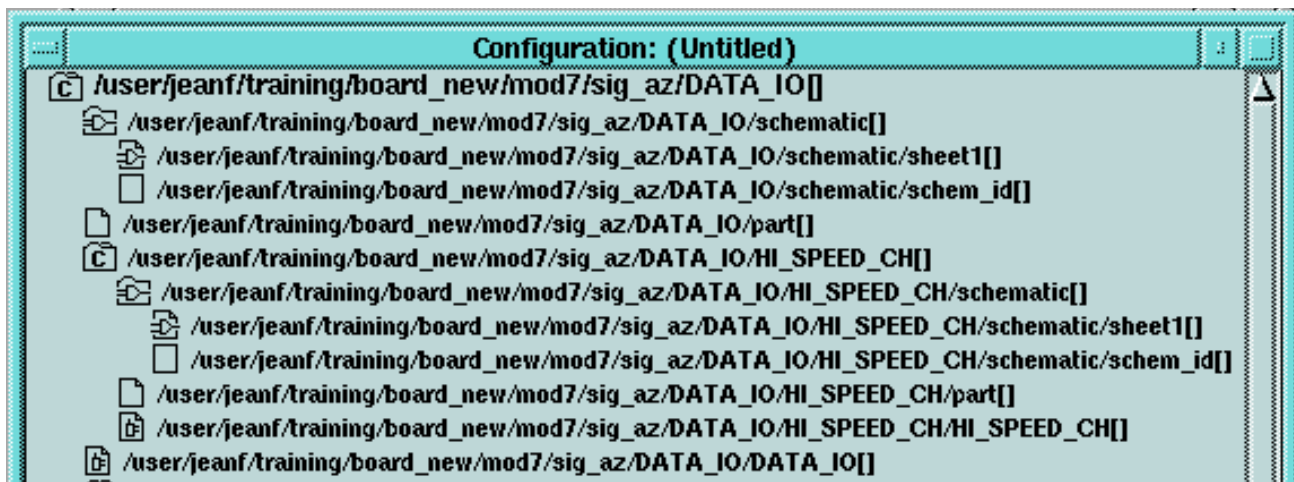
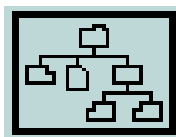


Figure 6-3. Design Manager Configuration Window



2. **Configuration object.** This design object, represented in directory listings by the icon on the left, records the exact contents of your configuration. It is not a directory and does not contain the design data; it is a record of the design objects in the configuration.



## Creating a Design Configuration

A directory is an example of a simple configuration. When a directory includes all elements of a design (no references outside of the directory), you do not need to create a design configuration; you can operate on the top-level directory.

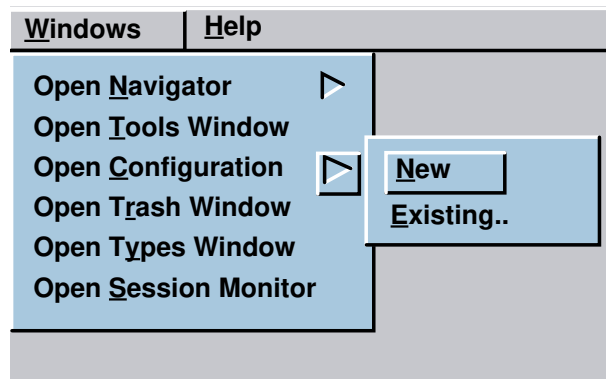
You need to create a configuration when your design references objects outside the design directory, or when the design directory contains items that you do not want to include in the configuration. The major steps to create a configuration are:

1. Open a new, untitled Configuration window.
2. Manually add primary entries to the configuration.
3. Specify build rules for including secondary entries.
4. Build the configuration. The Build function uses your build rules to traverse the containment hierarchy and, optionally, the reference network to identify and add secondary entries.
5. Save the configuration.

## Opening a Configuration Window

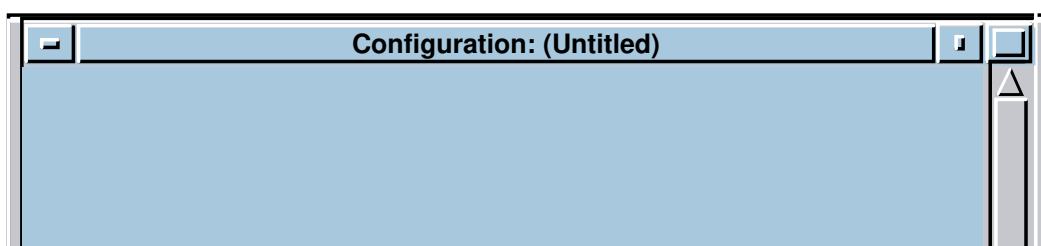


The Configuration window displays the names of design objects in the configuration that you are creating and modifying. In the Design Manager, you open a window for a new design configuration by either clicking the Select mouse button on the **Palette > Config** icon, shown at the left, or choosing the **Windows > Open Configuration > New** pulldown menu item, shown in [Figure 6-4](#).



**Figure 6-4. Design Manager Windows Menu**

The Design Manager displays a new, untitled Configuration window, shown in [Figure 6-5](#).



**Figure 6-5. Untitled Configuration Window**

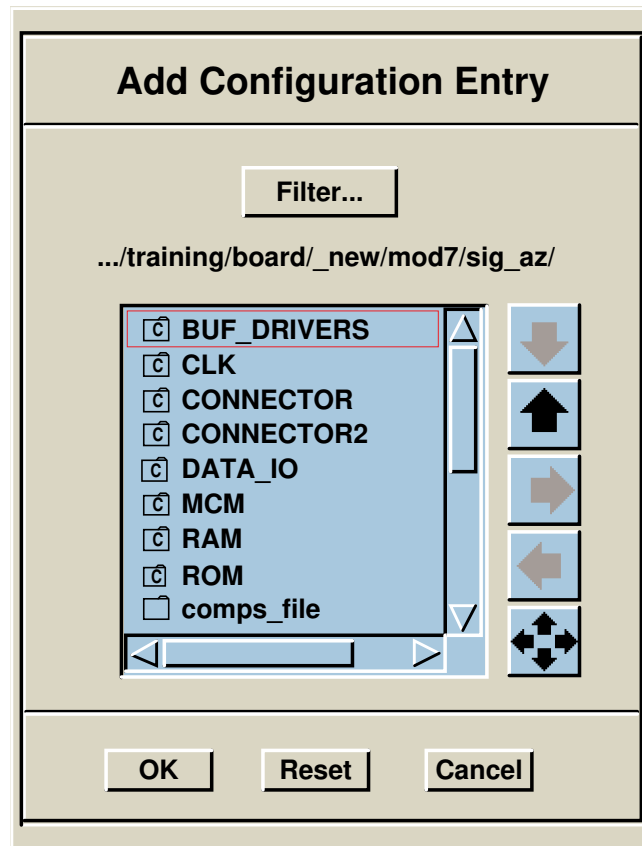
## Adding Primary Entries

A *primary entry* is a major part of a design, such as the *sig\_az* design directory you have been using in this training. *Secondary entries* are added to the configuration automatically because of their relationship to the primary entries.

For example, if *sig\_az* is a primary entry, *DATA\_IO* is a secondary entry because it is in the *sig\_az* directory. Configuration entries are not complete design objects; each entry is a single version of a design object. Because you can specify a particular version of an object, you can create a configuration of a previous version of a whole design.



You add a primary entry by clicking on the **Palette > Add Entry** icon, or by choosing the **[Configuration] Add Entry** popup menu item. The Add Configuration Entry dialog box, shown in [Figure 6-6](#), is displayed for you to specify which objects to add to the configuration.



**Figure 6-6. Add Configuration Entry Dialog Box**

You can also add a primary entry to a configuration by opening a navigator window in the Design Manager, navigating to the correct directory, and using the Select mouse button to drag an icon from the navigator window into the Configuration window. If you select multiple icons in the navigator window and drag one of the selected icons into the Configuration window, transparent images of all selected icons move together. When at least one of the icon images is in the Configuration window, release the Select mouse button.

When you add a primary entry, the icon and full pathname of the entry are displayed in the Configuration window. Square brackets at the end of the entry name indicate that the configuration entry is the current version of the object. If you want to add other versions of an object, choose the **[Configuration] Add Versions** popup menu item and specify the version in the dialog box.

## Setting Build Rules

After you add primary entries to the configuration, you specify build rules that the system uses to identify secondary entries. Default build rules for all primary entries include all design objects in the primary entry's containment hierarchy and reference network.

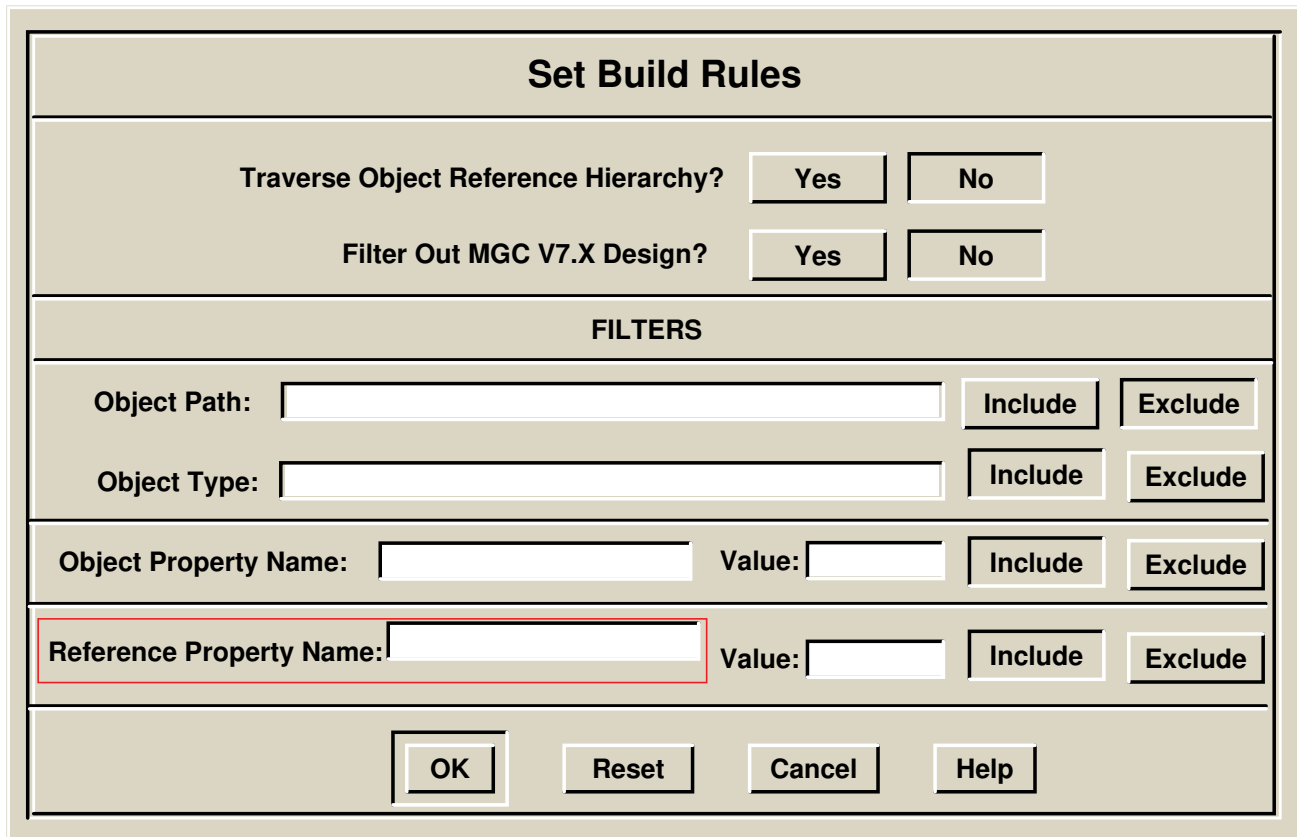
A design object's containment hierarchy includes that object and everything in the directory tree beneath it. You cannot disable containment traversal; however, you can specify filters in the build rules to exclude certain contained objects.

A reference network includes everything that can be reached by following an object's references. You can disable reference traversal in the build rules.

Each primary entry has its own set of build rules, even if you specify the same set of rules for multiple entries. You can change the build rules for a primary entry at any time.



To specify build rules, select one or more primary entries in the Configuration window using the Select mouse button. Click on the **Palette > Set Build Rules** icon, or choose the **[Configuration] Set Build Rules** popup menu item.



The image shows a 'Set Build Rules' dialog box. It has a title bar 'Set Build Rules'. Below the title bar, there are two rows of options. The first row is 'Traverse Object Reference Hierarchy?' with 'Yes' and 'No' buttons. The second row is 'Filter Out MGC V7.X Design?' with 'Yes' and 'No' buttons. Below these is a section titled 'FILTERS'. It contains four rows of filters. The first row is 'Object Path:' with a text box and 'Include' and 'Exclude' buttons. The second row is 'Object Type:' with a text box and 'Include' and 'Exclude' buttons. The third row is 'Object Property Name:' with a text box, 'Value:' with a text box, and 'Include' and 'Exclude' buttons. The fourth row is 'Reference Property Name:' with a text box, 'Value:' with a text box, and 'Include' and 'Exclude' buttons. At the bottom of the dialog box are four buttons: 'OK', 'Reset', 'Cancel', and 'Help'.

Set Build Rules			
Traverse Object Reference Hierarchy?		Yes	No
Filter Out MGC V7.X Design?		Yes	No
FILTERS			
Object Path:	<input type="text"/>	Include	Exclude
Object Type:	<input type="text"/>	Include	Exclude
Object Property Name:	<input type="text"/>	Value: <input type="text"/>	Include Exclude
Reference Property Name:	<input type="text"/>	Value: <input type="text"/>	Include Exclude
OK Reset Cancel Help			

**Figure 6-7. Set Build Rules Dialog Box**

Enter the following information in the Set Build Rules dialog box, shown in [Figure 6-7](#):

- **Reference Traversal.** You turn reference traversal on if you want all objects referenced by the design included in the configuration. Referenced objects, their contents (because containment traversal is always enabled), and everything referenced by those objects are added to the configuration as secondary entries. Traversal of references and containment hierarchies continues until all associated containers and references are explored.

When Reference Traversal is on, one version of each referenced object is included in the configuration. The version to be included depends upon whether the reference is current or fixed. For complete information about current and fixed references, refer to section "Specifying the Build Rules" in the [Design Manager User's Manual](#).

- **V7.X Filter.** Specify whether to include pre-V8 Mentor Graphics data in the design configuration.
- **Other Filters.** You can use four other filters to specify whether to include or exclude objects. (The **Include** and **Exclude** buttons at the end of the text entry boxes are not shown in [Figure 6-7](#).) When you type into any of the filter text fields, the dialog box automatically expands, allowing you to specify more filter patterns for that category.

The four filter categories work together to filter potential secondary entries out of the configuration as follows:

- In any filter category, you can combine filters that include and exclude objects.
- If a design object passes (is not filtered out by) each of the four filter categories, the object is added to the configuration.
- Design objects always pass empty (unspecified) filter categories.

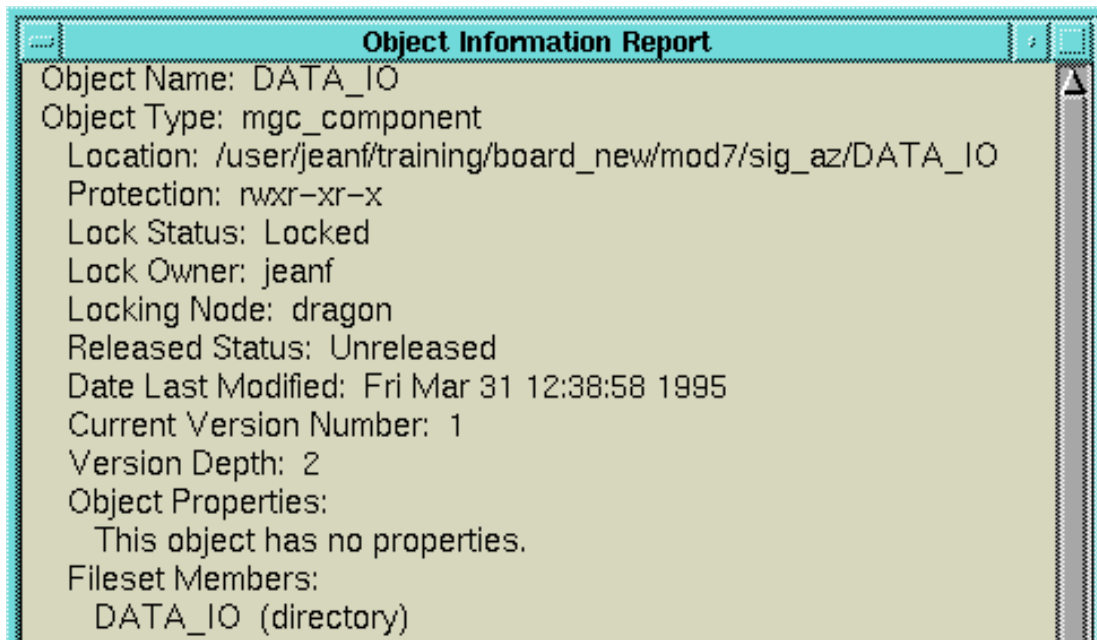
**Object Path.** You can enter an object pathname to include or exclude in the configuration. You can use UNIX regular expressions as wildcards in the Object Path text field.

For information about UNIX regular expressions, refer to the [Design Manager User's Manual](#).

**Object Type.** You can enter an object type as a filter to include or exclude. You can use wildcards in this text field.



To determine the type of a design object, select the object in a navigator window and either click on the **Palette > Report Info** icon, or choose **Report > Object Info** from the pulldown menu or the navigator window popup menu. [Figure 6-8](#) shows an example of the information provided.



**Figure 6-8. Object Information Report Window**

**Object Property.** This filter checks property names and, optionally, the property values of all containment objects to determine if they are included in the configuration. You cannot use wildcards in this category.

**Reference Property.** This filter checks property names and, optionally, the property values of all objects reached through references to determine if they are included in the configuration. You cannot use wildcards in this category.

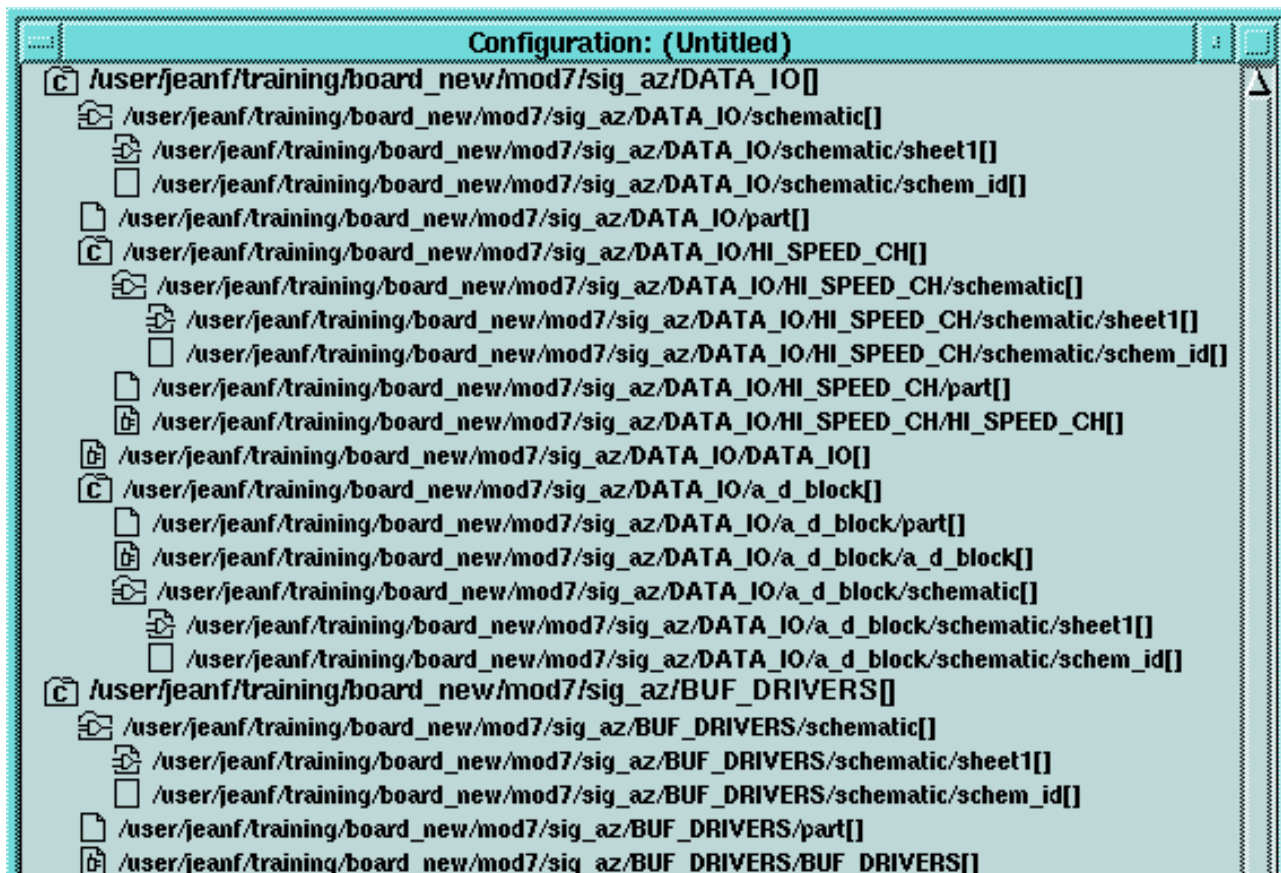


## Building the Configuration



After you define build rules for each primary entry, you can build the configuration by clicking on the **Palette > Build** icon, or by choosing the **[Configuration] Build** popup menu item.

Design Manager identifies and adds secondary entries by following the build rules you specified for each primary entry. [Figure 6-9](#) shows the Configuration window after a build.



**Figure 6-9. Configuration Window After a Build**

If the Design Manager encounters an error during the build, the operation does not terminate. The operation continues to build as much of the configuration as possible, and displays error messages for entries that fail.

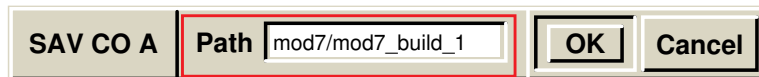
The build process automatically resolves any target conflicts that exist in the configuration. Conflict resolution occurs after the normal build process is complete.

You can redefine your build rules and build the configuration several times to get the exact configuration you like.

## Saving the Configuration

After you build the configuration and are satisfied that it contains the entries that you want, you can save it for later use. If you do not save a configuration, it is discarded when you exit the Configuration window.

To save an untitled configuration, choose the **[Configuration] Save As** popup menu item. The prompt bar shown in [Figure 6-10](#) is displayed for you to enter a pathname for the configuration. For example, enter the pathname to your design directory, and then append a name for the configuration object.



**Figure 6-10. Save Configuration As Prompt Bar**

The name you specify for the configuration is displayed in the title bar of the Configuration window, and the configuration object just saved is automatically added to the configuration.

## Releasing a Design Configuration



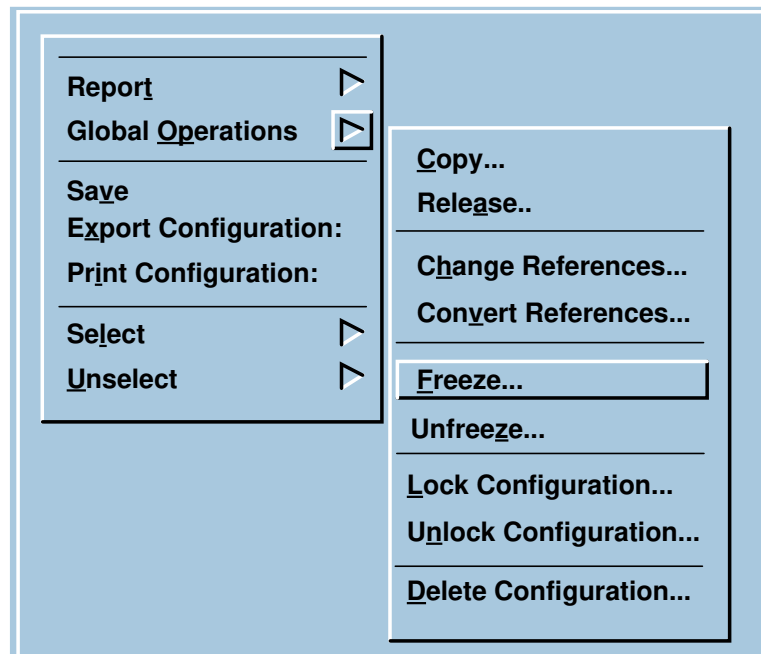
Releasing a design creates a protected copy of each object in the configuration. The Design Manager assigns the "released" attribute to every versioned object in the release directory, and does not allow you to create new versions of these released objects. If you want to open and evolve released, versioned design objects, you must copy the released configuration to another directory. The *released* attribute only prevents you from evolving versioned objects. It does not prevent you from opening an unversioned design object, or from deleting or moving any released design object.

To release a design configuration, perform the following steps:

1. If the configuration is untitled, save it by choosing the **[Configuration] Save As** popup menu item.

Saving the configuration automatically includes the configuration object in the configuration.

2. Lock the configuration by choosing the **[Configuration] Global Operations > Lock Configuration** popup menu item.



**Figure 6-11. [Configuration] Global Operations Submenu**

Locking the configuration is optional, but recommended, because it prevents anyone else from modifying or deleting configuration entries while you perform the release.

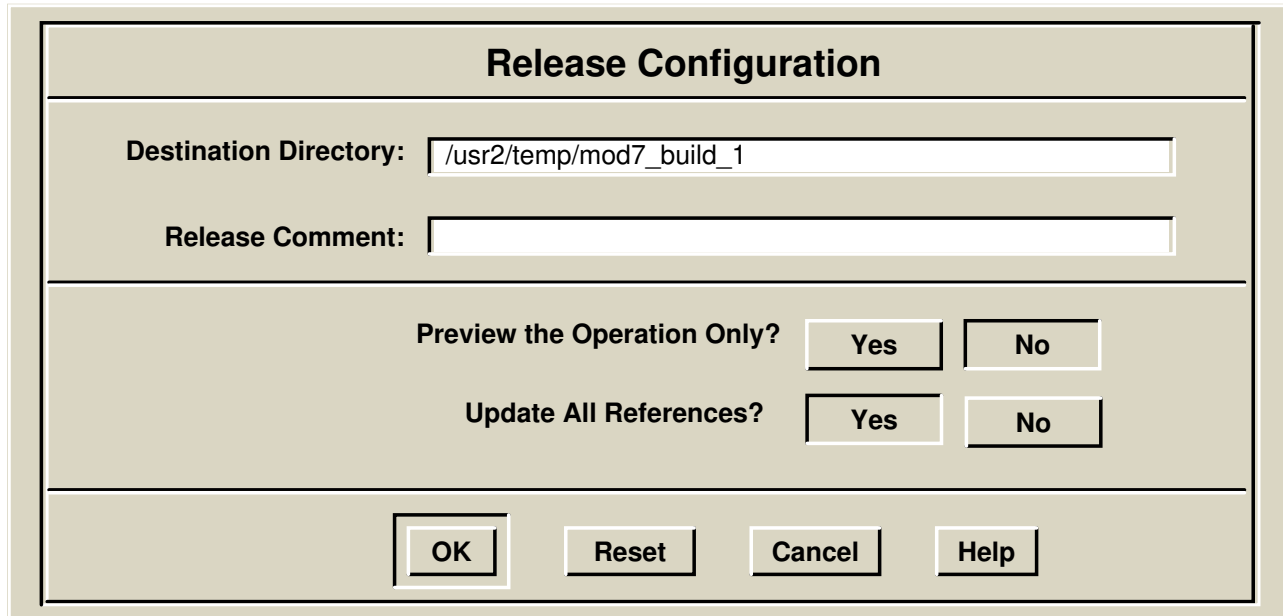
3. Click the **Palette > Build** icon to update the display to exactly reflect the contents of the configuration.



This step is also optional, but is recommended in case someone moved or deleted a configuration entry. If you locked the configuration, then when you build it all secondary entries are locked as they are added to the configuration.

4. Click on the **Palette > Release** icon to display the dialog box shown in [Figure 6-12](#).





The image shows a 'Release Configuration' dialog box with a light beige background and a dark border. At the top, the title 'Release Configuration' is centered in a bold, black font. Below the title, there are two text entry fields. The first is labeled 'Destination Directory:' and contains the text '/usr2/temp/mod7\_build\_1'. The second is labeled 'Release Comment:' and is currently empty. Below these fields, there are two sets of buttons. The first set is for 'Preview the Operation Only?' with 'Yes' and 'No' buttons. The second set is for 'Update All References?' with 'Yes' and 'No' buttons. At the bottom of the dialog, there are four buttons: 'OK', 'Reset', 'Cancel', and 'Help', arranged horizontally.

**Figure 6-12. Release Configuration Dialog Box**

5. Enter the pathname where you want each configuration entry to be released.

If the destination directory does not exist, then, when you execute the Release Configuration dialog box, another dialog box appears, asking if you want to create the directory.

6. You can optionally assign a comment by typing in the **Release Comment** text entry field.

The comment is added to the configuration object as the value of the Version property.

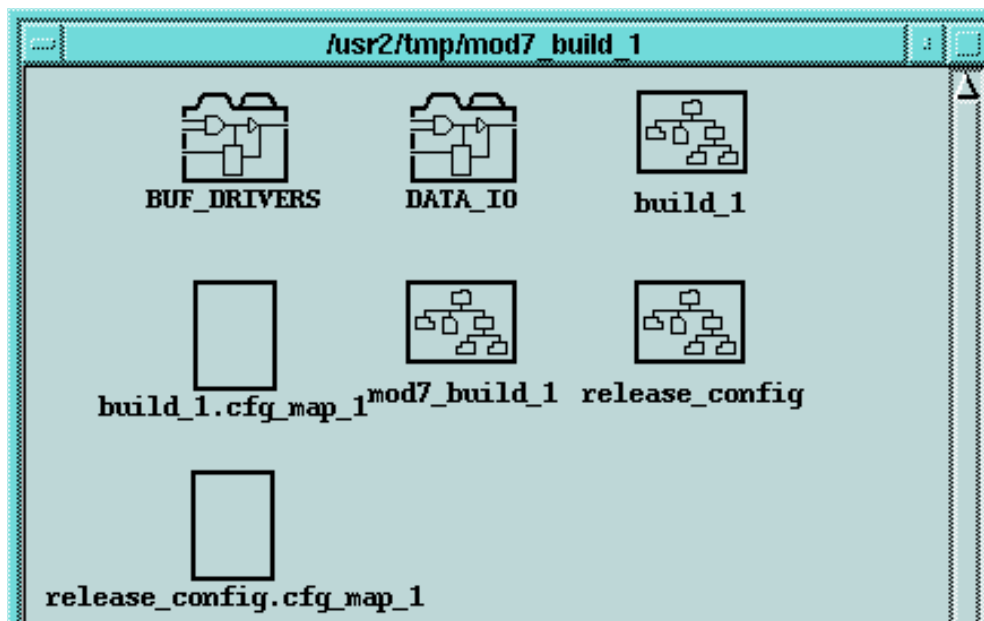
7. Specify whether to **Preview the Operation Only?** by clicking on the button.

If you specify **No**, Design Manager attempts to release the configuration. If you specify **Yes**, an information window appears and reports the **from** and **to** paths of each configuration entry and any naming conflicts between existing and released objects.

8. Specify whether to **Update All References?** by clicking on the button, and **OK** the dialog box.

If you specify **No**, the references of the released objects continue to point to the original location. If you specify **Yes**, the references of the released objects point to their targeted objects at their new destinations.

Figure 6-13 shows the newly released design objects in a navigator window.



**Figure 6-13. Released Design Directory**

A single version of each design object is created in the destination directory. All versioned design objects are initially released to the destination directory as version 1, and subsequent releases to the same directory are layered on top of existing versions and are numbered sequentially.

The major difference between releasing and copying is that the Release function does not always overwrite conflicting design objects (objects having the same name) in the destination directory.

For versioned design objects, Release merges the conflicting objects by overlaying the new version on top of the older version and assigning the next higher version number.

In case of conflicts with unversioned objects, the unversioned object is overwritten. Unversioned containers, except for those of object type *mgc\_container*, which are ordinary directories, not only overwrite the container object, but also all of the container's contents.

## Operations on Design Configurations and Objects

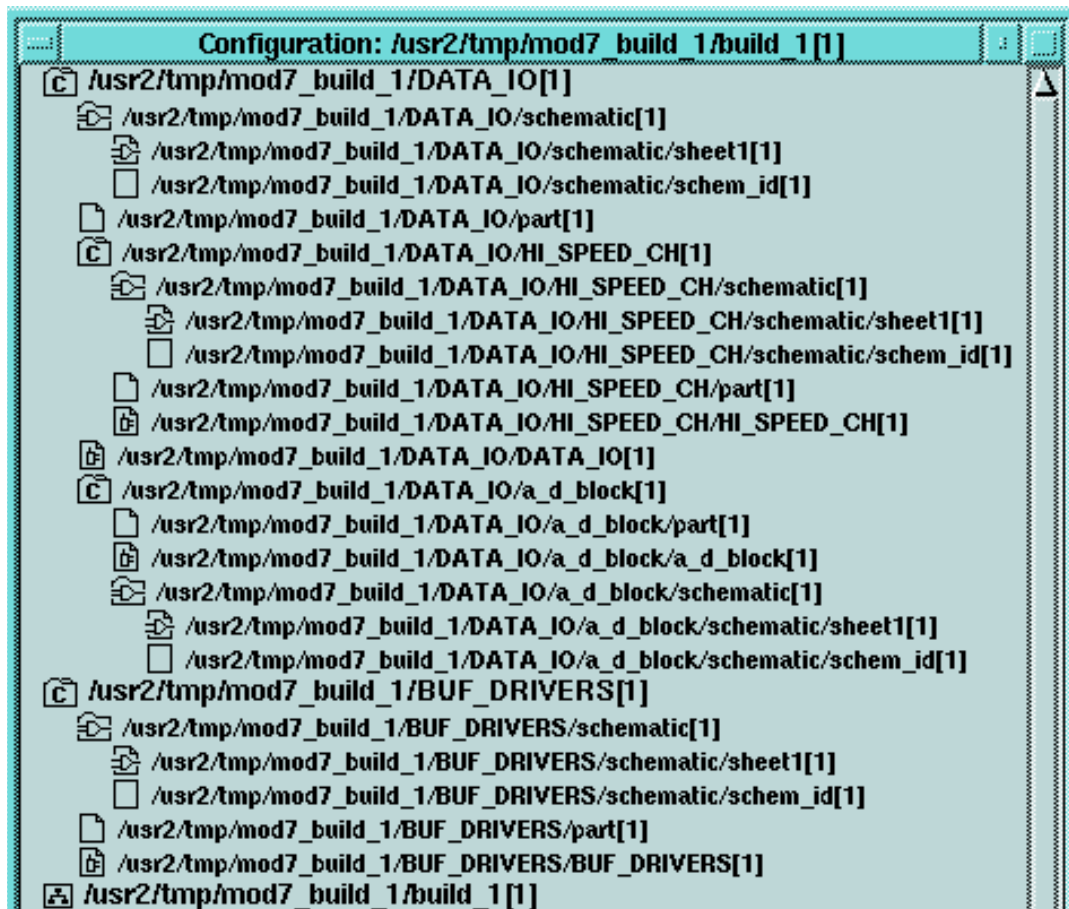
This section covers operations on design configurations including viewing, copying, freezing and unfreezing both design objects and design configurations, deleting a design configuration, and archiving a design.

### Viewing a Design Configuration

You can view a design configuration by containment hierarchy or by primary/secondary hierarchy.

#### Viewing the Containment Hierarchy

Viewing the containment hierarchy is the default mode for the Configuration window. This mode is similar to listing a file system directory and subdirectories: it uses successive levels of indentation to show which objects are contained by other objects. The containment view does not show the primary and secondary relationship of the configuration entries. [Figure 6-14](#) shows an example of a Configuration window containing entries of a containment hierarchy.



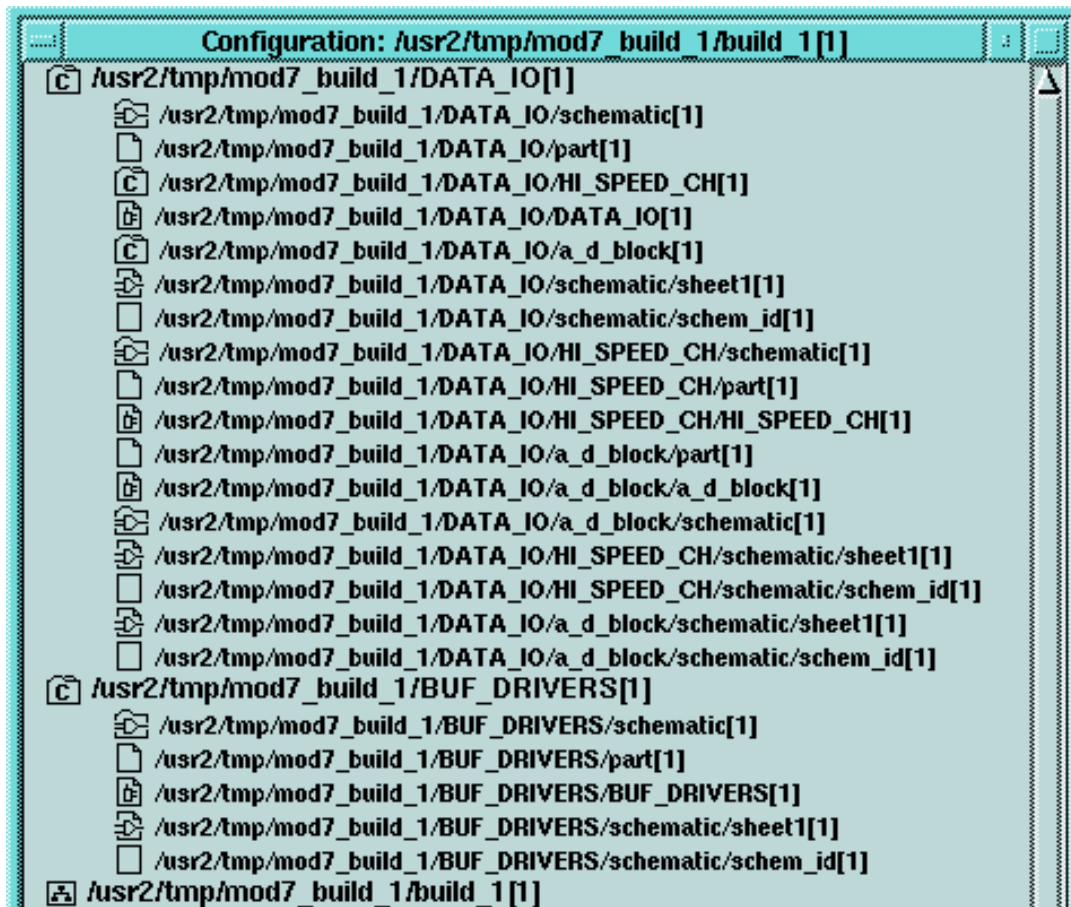
**Figure 6-14. Viewing the Containment Hierarchy**

When you copy or release a design configuration, you might want to specify a different target path for some configuration entries than the copy or release destination. Containment viewing is helpful when you need to identify which objects are *retargetable*. A retargetable object is one whose parent directory is not in the configuration; therefore, only left-justified objects in a containment view are retargetable.



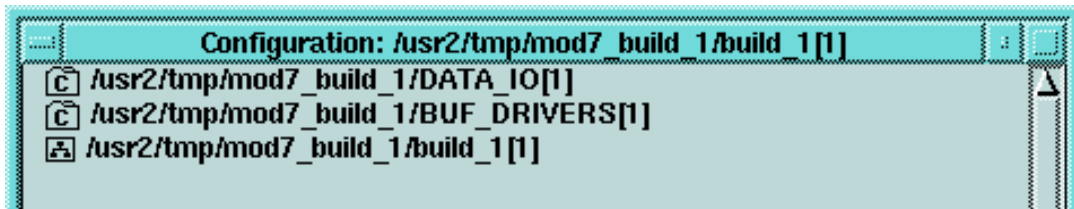
## Viewing the Primary and Secondary Hierarchy

You can view the primary and secondary design relationships in your configuration by choosing the **[Configuration] View Primaries** popup menu item. As shown in [Figure 6-15](#), the primary entries are left-justified, and each primary has its secondary entries indented beneath it.



**Figure 6-15. Viewing the Primary and Secondary Hierarchy**

To see only the primary entries, choose **[Configuration] Hide Secondaries**. [Figure 6-16](#) shows only the primary entries in the Configuration window.



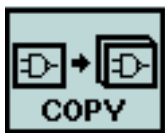
**Figure 6-16. Viewing Only Primary Entries**

To restore the view of both primaries and secondaries, choose the **[Configuration] View Secondaries** popup menu item.

To view the containment hierarchy again, choose the **[Configuration] View Containment** popup menu item.

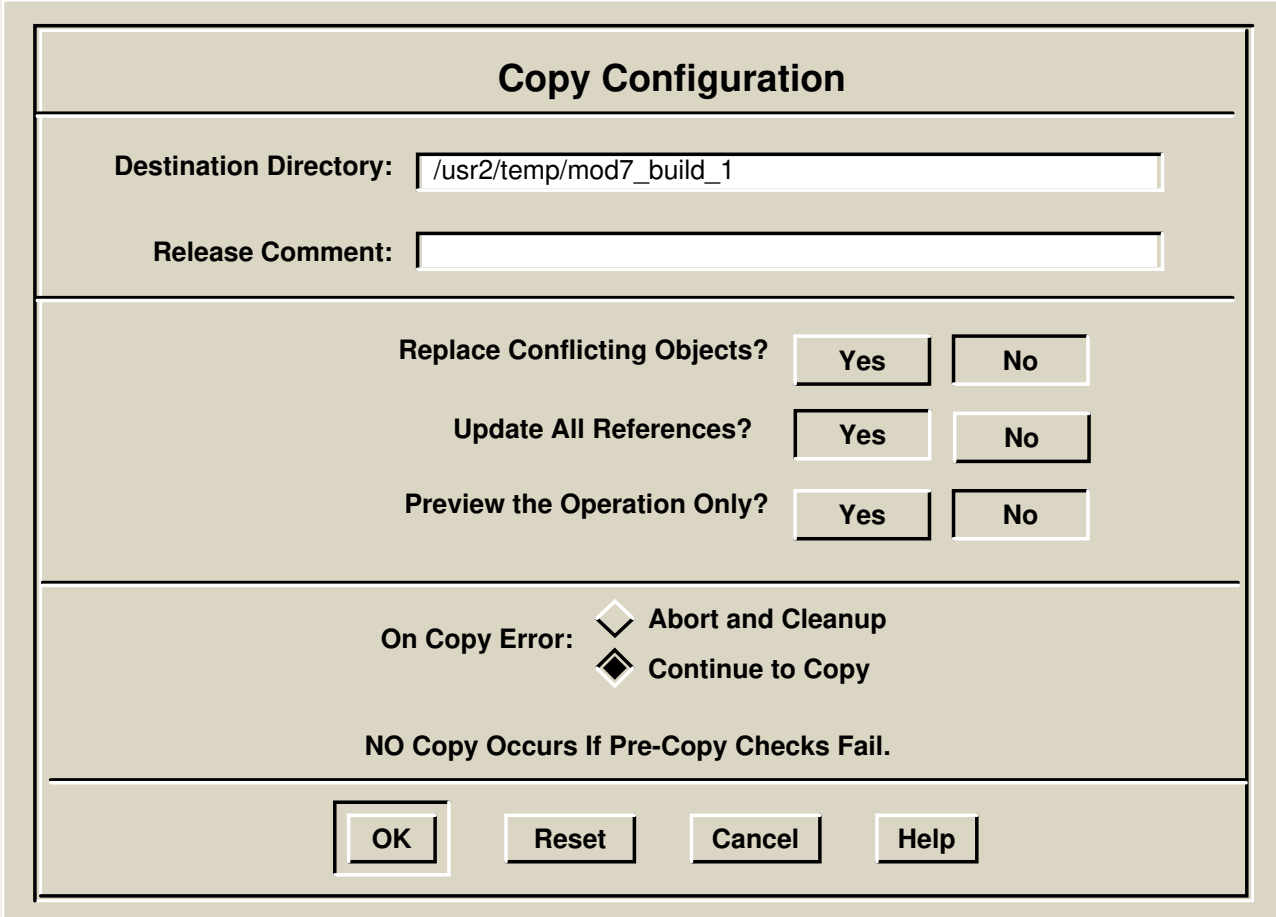
## Copying a Configuration

The Copy Configuration function works like the Copy Object function, except that it operates only on each object in the configuration, rather than every object in a directory.



To copy a configuration, be sure the Configuration window is active, then click the Select mouse button on the **Palette > Copy** icon. The dialog box shown in [Figure 6-17](#) is displayed for you to specify a destination pathname.

You can specify whether to replace objects having the same name, whether to update references, and what to do in case of an error. You can also preview the copy before actually copying the configuration.



The image shows a 'Copy Configuration' dialog box with a light beige background and a dark border. At the top, the title 'Copy Configuration' is centered in a bold font. Below the title, there are two input fields: 'Destination Directory:' with the text '/usr2/temp/mod7\_build\_1' and 'Release Comment:' which is empty. These fields are separated by a horizontal line. Below the line, there are three rows of options, each with a label and two buttons ('Yes' and 'No'). The first row is 'Replace Conflicting Objects?', the second is 'Update All References?', and the third is 'Preview the Operation Only?'. Another horizontal line follows. Below this line, there is a section for 'On Copy Error:' with two radio button options: 'Abort and Cleanup' (which is selected) and 'Continue to Copy'. Below this, the text 'NO Copy Occurs If Pre-Copy Checks Fail.' is displayed. At the bottom of the dialog, there is a row of four buttons: 'OK', 'Reset', 'Cancel', and 'Help'.

Copy Configuration	
Destination Directory:	/usr2/temp/mod7_build_1
Release Comment:	
Replace Conflicting Objects?	<input type="button" value="Yes"/> <input type="button" value="No"/>
Update All References?	<input type="button" value="Yes"/> <input type="button" value="No"/>
Preview the Operation Only?	<input type="button" value="Yes"/> <input type="button" value="No"/>
On Copy Error:	<input checked="" type="radio"/> Abort and Cleanup <input type="radio"/> Continue to Copy
NO Copy Occurs If Pre-Copy Checks Fail.	
<input type="button" value="OK"/> <input type="button" value="Reset"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>	

**Figure 6-17. Copy Configuration Dialog Box**

When you copy a configuration, each object is copied into the destination that you specify. If the configuration includes a container or directory, that container or directory is copied, but only the contents shown in the configuration are also copied.

## Freezing a Design Object

As you create new versions of a design object, the versions that exceed the object's version depth are pruned (deleted) automatically. You can prevent versions from being pruned by freezing them. You can freeze a version of a single object, or all objects in a design configuration. Freezing a version does not prevent you from deleting the entire design object, including the frozen version, while in the navigator.

If a newly unfrozen version is outside of the design object's version depth, then when you create another version, the unfrozen version is deleted.

To freeze and unfreeze a version, perform the following steps:

1. In a Design Manager navigator window, click on the object whose version you want to freeze, and choose the **[Navigator] Report > Show Versions** popup menu item. You need to select the actual versioned object, as shown in [Figure 6-18](#).

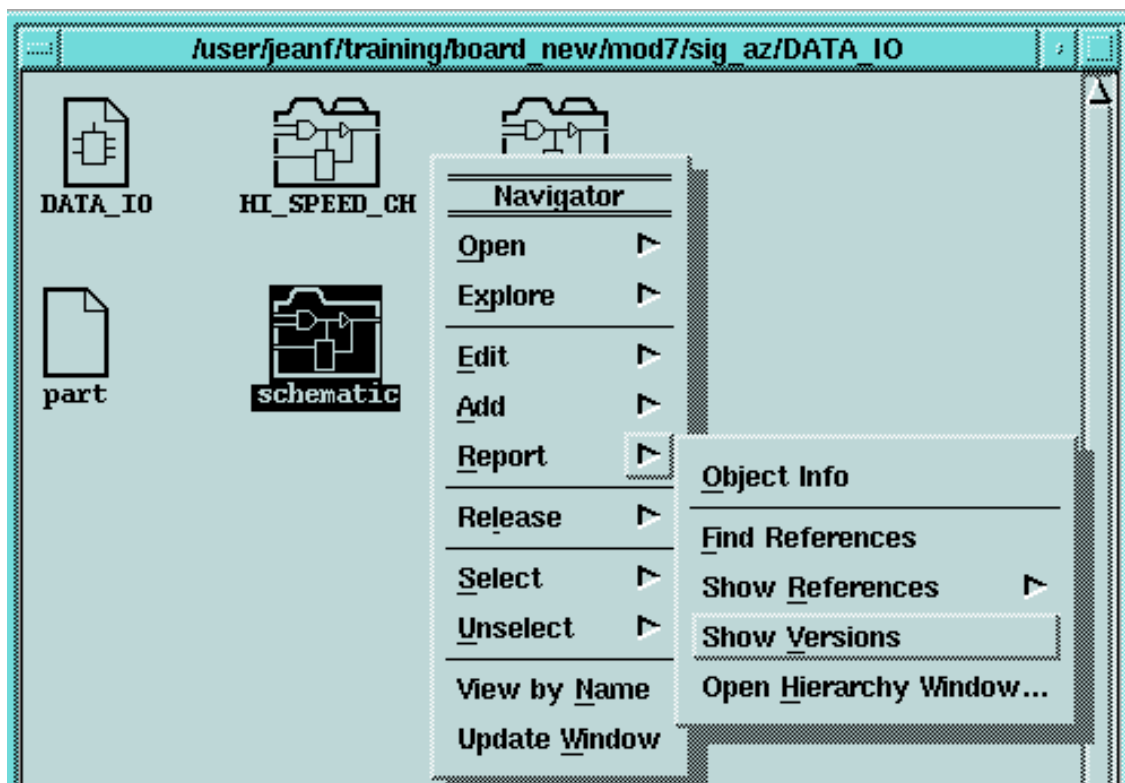
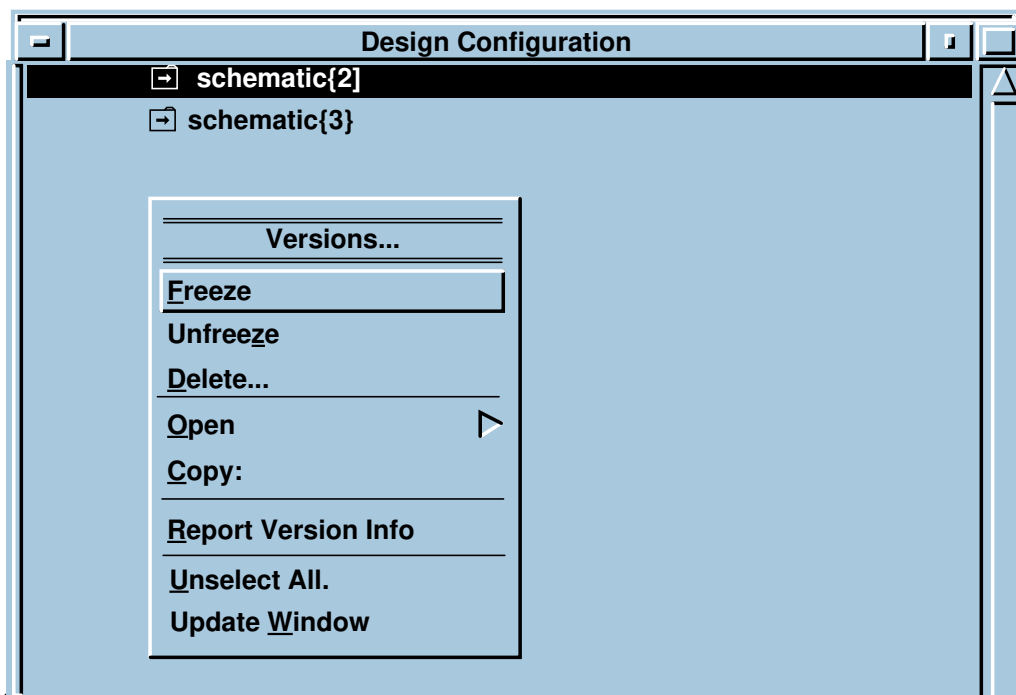


Figure 6-18. [Navigator] Report > Show Versions Menu

A Sequence Versions window is displayed in which the versions of the selected design object are listed, as shown in [Figure 6-19](#).



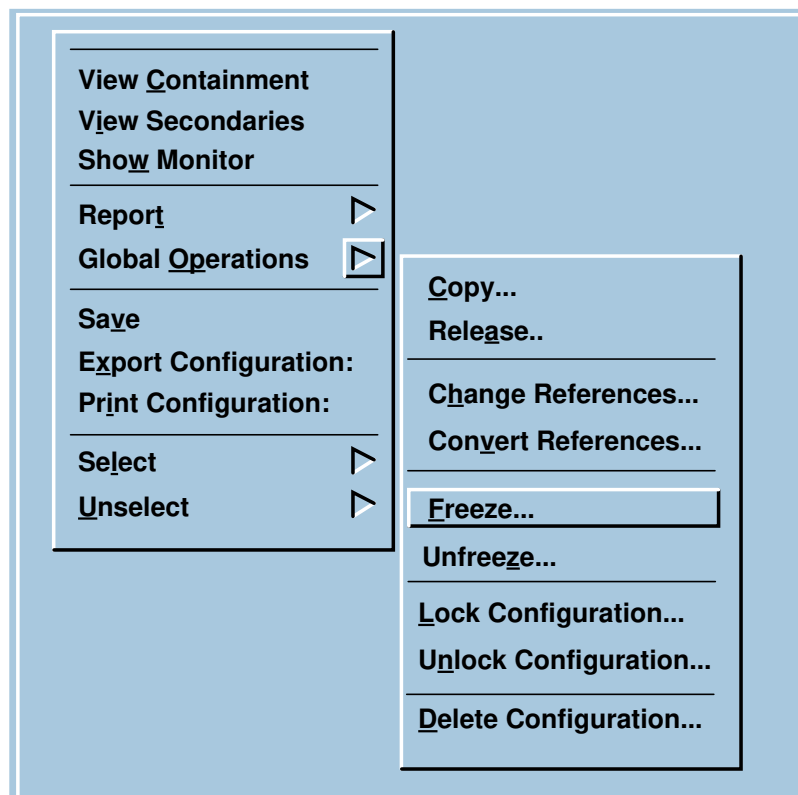
**Figure 6-19. Sequence Versions Window and Popup Menu**

2. To freeze a version, select the version in the Sequence Versions window, and choose the **Versions > Freeze** popup menu item.
3. To unfreeze a frozen version, choose the **Versions > Unfreeze** popup menu item.

## Freezing a Design Configuration

You can freeze versions of all objects within a design configuration to prevent the Design Manager version depth mechanism from deleting versions. Only versioned objects can be frozen and unfrozen.

To freeze an entire design configuration, activate the Configuration window and choose the **[Configuration] Global Operations > Freeze** popup menu item. This menu is shown in [Figure 6-20](#).



**Figure 6-20. [Configuration] Global Operations > Freeze**

## Deleting a Design Configuration

After you build a design configuration, you can delete the design object versions represented in the configuration by deleting the configuration. This deletes just the version in the configuration, not all versions of the design object.

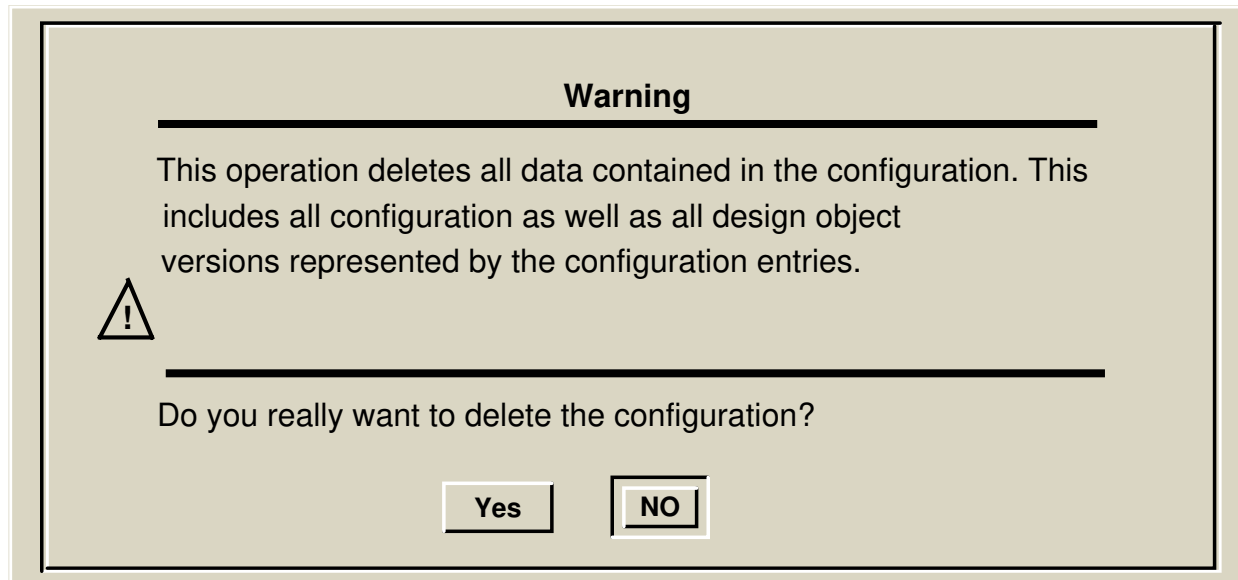
For example, if design object *data\_io* has versions 3, 5, 7, and 8, and if the configuration includes version 5, deleting the configuration leaves *data\_io* with versions 3, 7, and 8.



*Because unversioned design objects have only one version, deleting a configuration that contains unversioned objects deletes these objects entirely.*

You can interrupt a configuration deletion by pressing the Kill key; however, this does not return the design configuration to its original state. Any objects in the configuration that were deleted prior to the interrupt are permanently deleted; all others are not deleted.

To delete a design configuration, choose the **[Configuration] Global Operations > Delete Configuration** popup menu item. The message box shown in [Figure 6-21](#) is displayed so you do not mistakenly delete data.



**Figure 6-21. Delete Configuration Message Box**

## Archiving and Restoring Designs

After you release your configuration, you can archive the configuration in its current form for later use. To archive or restore a configuration, the following assumptions must be true:

- You are familiar with both Design Manager and the configuration.
- You are using the same platform environment for both source and destination. This means the following statements are true:
  - The platform type and operating system version are compatible.
  - The Mentor Graphics software tree version and tools version are the same.
  - The required type registries and fonts are the same.
- You know which objects need to be archived, including all libraries outside of the design configuration.



## Archiving a Configuration

To archive your configuration, perform the following steps:

1. Create a new configuration.
2. Save the configuration.
3. Use the **[Navigator] Edit > Copy** or **[Navigator] Edit > Release** popup menu item to copy or release the configuration to an empty directory.
4. Archive the directory from step 3, using the `tar` or `cpio` shell command.

For information about using either of these shell commands, refer to their online command reference page by typing one of the following commands in a Bourne or C shell:

```
man tar
man cpio
```

## Restoring a Configuration

To restore a design configuration, perform the following steps:

1. Restore the archived directory using the `tar` or `cpio` command.

For information about using either of these shell commands, refer to their online command reference page by typing one of the following commands in a Bourne or C shell:

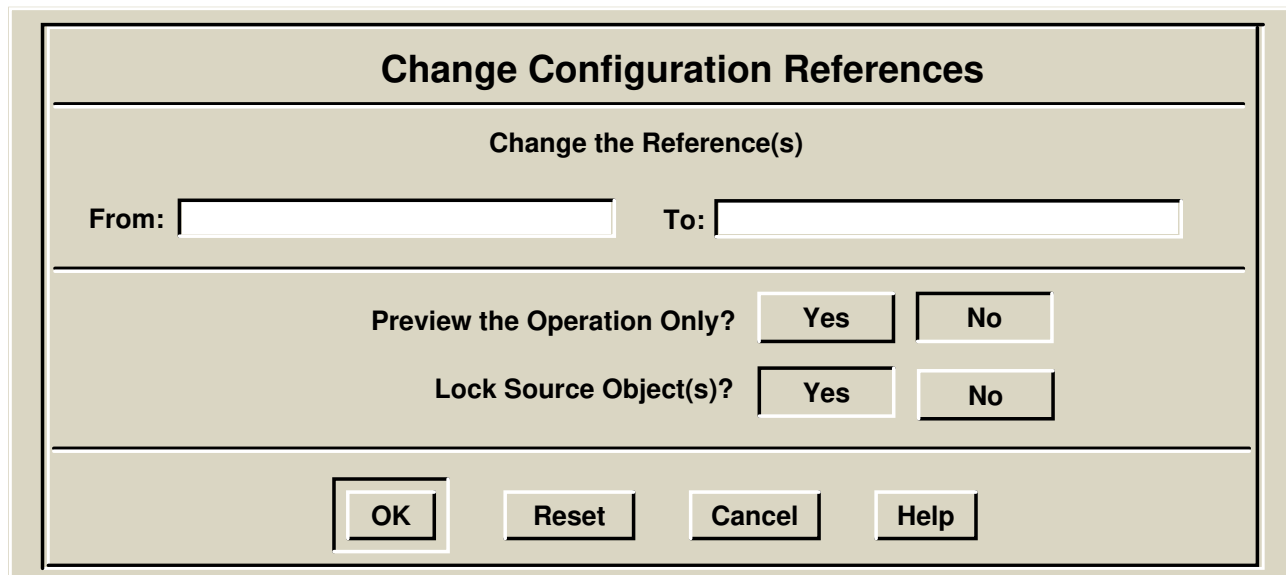
```
man tar
man cpio
```

2. Note the pathname of the configuration in its new location.
3. Update references within the configuration that point to other objects within the configuration to the new pathname you noted in step 2.
4. Update references within the configuration that point to objects not in the configuration.

## Updating Restored References

You can change references within an entire configuration. The pathname you specify to change is compared to all entries in the configuration; all entries that match are changed to the new pathname you specify.

To update references within an archived configuration that you have restored, choose the **[Configuration] Global Operations > Change References** popup menu item. [Figure 6-22](#) shows the Change Configuration References dialog box in which you specify the original reference pathnames and new pathnames.



**Figure 6-22. Change Configuration References Dialog Box**

When you begin typing, the dialog box expands so you can enter more than one pair of pathnames. You can use wildcards in the **From** text entry field. The Change Configuration References function does not check to see if the resulting reference points to an existing object.

The safest way to change references in a configuration is to preview the change first. Specifying **Preview Operation Only? Yes** causes the function to report the expected results to the monitor window. If the report accurately reflects your intended actions, you can execute the function again, specifying **No**, and then the references are changed.

If you specify **Lock Source Object(s)**? **Yes**, no one can change any of the entries that make up the configuration while you are changing references.

If you halt this operation, the configuration is not returned to its original state. Any changes made to reference pathnames of a configuration entry prior to the interruption are permanent, while reference pathnames that have not yet been processed remain unaffected.

## Lab Exercise

In this lab exercise, you configure a design for releasing and archiving. Upon completion of this lab exercise you are able to:

- Configure a design.
- Release a design.
- Restore a design.
- Change design references.

Turn to Module 7—Lab 6: "Releasing a PCB Design".

# Lab 6

## Releasing a PCB Design

### Introduction

In this lab exercise, you use Design Manager to create a design configuration. Upon completion of this lab exercise you are able to:

- Configure a design.
- Release a design.
- Restore a design.
- Change design configurations.

### Procedure

Use Design Manager to create a design configuration.

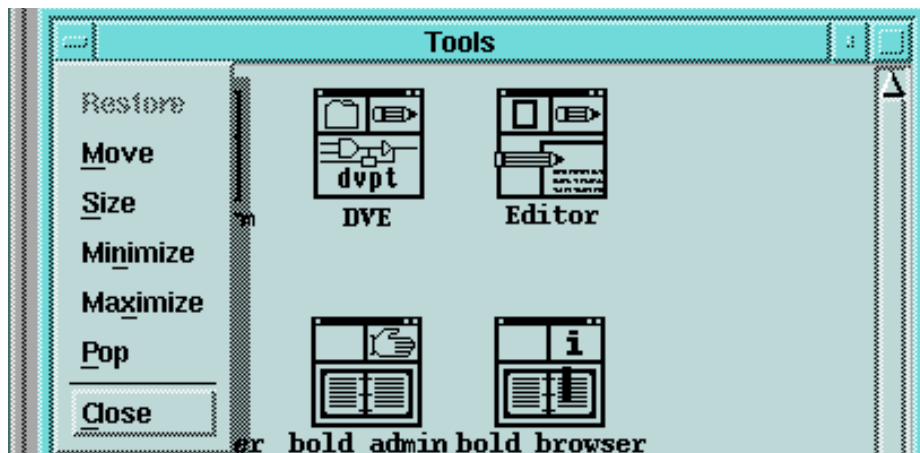
## Preparation for Lab

1. Invoke Design Manager from a shell, if it is not displayed:

```
$MGC_HOME/bin/dmgr
```

By default, Design Manager displays a Navigator window and a Tools window. You do not need the Tools window in this lab.

2. Close the Tools window by choosing **Close** from the **Windows** pulldown menu in the upper-left corner of the Tools window.



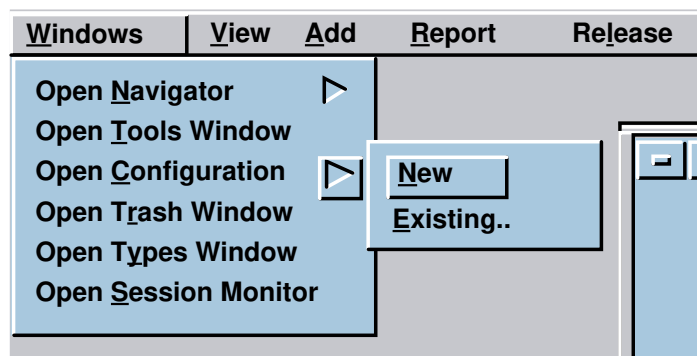
3. Navigate to your **sig\_az** design, and display the contents. The pathname is:

```
your_path/training/board_new/mod7/sig_az
```

## Creating a Design Configuration

In this part of the lab, you open a new configuration window, add primary entries, define build rules, and build the configuration. You look at both the containment hierarchy and the design hierarchy, define new build rules, and build and save the configuration.

1. Choose the **Windows > Open Configuration > New** pulldown menu item.



This displays a new, untitled Configuration window.

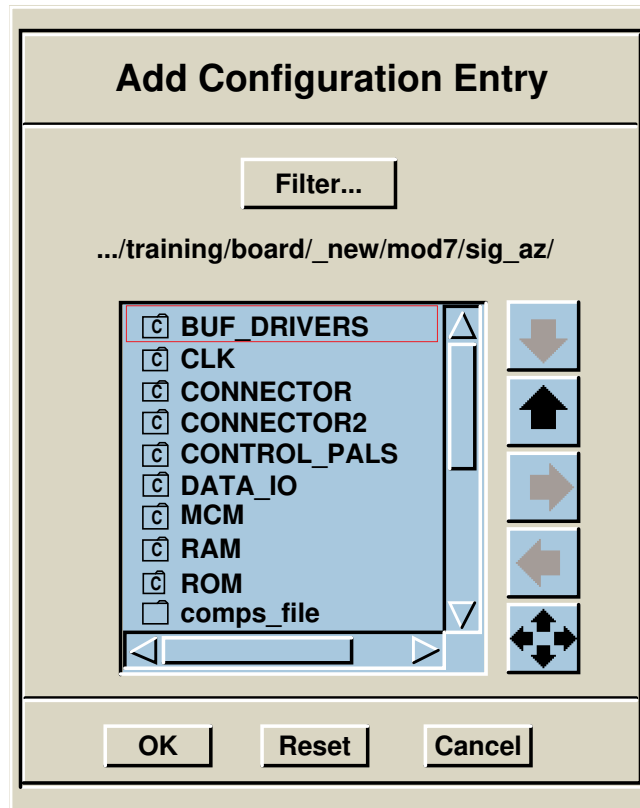
2. In the Navigator window in which **sig\_az** displays, move the cursor to the **DATA\_IO** component icon. Press and hold the Select mouse button as you drag a ghost image of the icon into the Configuration window. Release the mouse button.

The complete pathname of the DATA\_IO component displays in the Configuration window as a primary entry.



3. Activate the Configuration window and click on the **Palette > Add Entry** icon.

The Add Configuration Entry dialog box displays.



4. Click the Select mouse button on **BUF\_DRIVERS** and **OK** the dialog box.

You now have two primary entries in the Configuration window. The only reason for adding the primary entries using different methods was to remind you that there are different methods for most tasks.

Select and define build rules for the **DATA\_IO** entry, and use the default build rules for **BUF\_DRIVERS**.



5. In the Configuration window, click on the **DATA\_IO** pathname, then click on the **Palette > Set Build Rules** icon.
6. In the Set Build Rules dialog box, enter an **Object Path** for *gen\_lib* components, then click on the **Exclude** button at the end of the text entry field, and **OK** the dialog box. The pathname is:

**.../mod7/gen\_lib/\***



Set Build Rules			
Traverse Object Reference Hierarchy?	<input type="button" value="Yes"/>	<input type="button" value="No"/>	
Filter Out MGC V7.X Design?	<input type="button" value="Yes"/>	<input type="button" value="No"/>	
FILTERS			
Object Path:	<input type="text" value="/user/jean/training/board_new/m07/gen_lib"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Type:	<input type="text"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Property Name:	<input type="text"/>	Value: <input type="text"/>	<input type="button" value="Include"/> <input type="button" value="Exclude"/>
Reference Property Name:	<input type="text"/>	Value: <input type="text"/>	<input type="button" value="Include"/> <input type="button" value="Exclude"/>
<input type="button" value="OK"/> <input type="button" value="Reset"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>			



- Click the Select mouse button on the **Palette > Build** icon.

Design Manager uses the rule you set for **DATA\_IO** and the default rules for **BUF\_DRIVERS** to identify and add secondary entries to the design configuration. Notice how many more secondary entries are listed for **BUF\_DRIVERS** than for **DATA\_IO**. This is because components in *gen\_lib* that are referenced by **BUF\_DRIVERS** are included.

If Design Manager finds errors during the build, the operation continues and a message displays when the build is complete. You can press the Show Monitor function key to read the transcript. For this design, you can ignore the error messages about missing technology files.

## Viewing the Configuration

The view in the Configuration window is by containment hierarchy, by default. You are going to change the view to design hierarchy and back to containment hierarchy to observe the differences.

1. In the Configuration window, choose the **[Configuration] View Primaries** popup menu item.

It is useful to look at the design hierarchy to determine which design objects you want in the configuration. For example, you do not need to keep the *parts* and *gen\_lib* directories with the configuration. Or, you can keep them, because you are archiving a design and need to restore the complete design at a future time.

2. Choose the **[Configuration] Hide Secondaries** popup menu item.

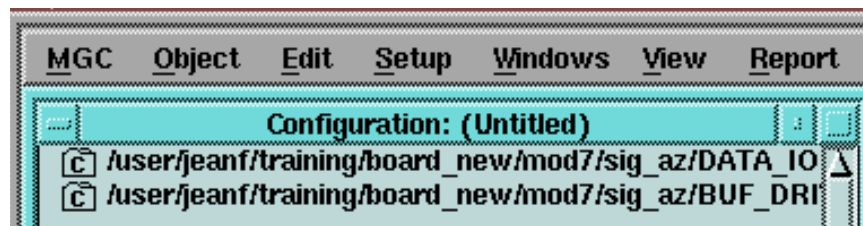
Only the two primary entries are displayed.

3. Choose the **[Configuration] View Containment** popup menu item.

Now the containment hierarchy displays for all primary and secondary entries. Next, you select both primary entries to reset the build rules.

4. Choose the **[Configuration] View Primaries** popup menu item.

Only primary entries are displayed in the Configuration window.



5. Select both primary entries in the Configuration window by pressing and holding the Select mouse button as you drag the cursor over both primary entries.

## Resetting Build Rules

1. With both primary entries selected in the Configuration window, click on the **Palette > Set Build Rules** icon.

The Set Build Rules dialog box displays for you to specify rules to identify all secondary entries.

2. Complete the dialog box:

Configuration: (Untitled)

g/board\_new/mod7/sig\_az/DATA\_IO

g/board\_new/mod7/sig\_az/BUF\_DRIVERS

### Set Build Rules

Traverse Object Reference Hierarchy?

Filter Out MGC V7.X Design Data?

#### FILTERS

Path: /user/jeanf/training/board\_new/mod7/parts/\*

Path: /user/jeanf/training/board\_new/mod7/gen\_lib/\*

3. Click the **Palette > Build** icon.

The build operation is much faster now, because Design Manager is not traversing any libraries.

4. In the Configuration window, choose the **[Configuration] View Secondaries** popup menu item.

This displays the entire configuration, instead of only the primaries.



5. Select the two primary entries once more and click on the **Palette > Set Build Rules** icon.

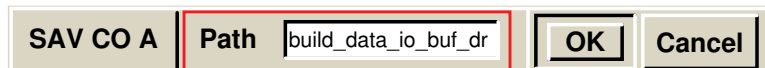


6. In the Set Build Rules dialog box, exclude only the components in **.../mod7/gen\_lib**, **OK** the dialog box, and click on the **Palette > Build** icon.

You are going to save this configuration and create another one.

7. Choose the [**Configuration**] **Save As** popup menu item.
8. In the prompt bar that appears, enter:

**...mod7/build\_data\_io\_buf\_dr**



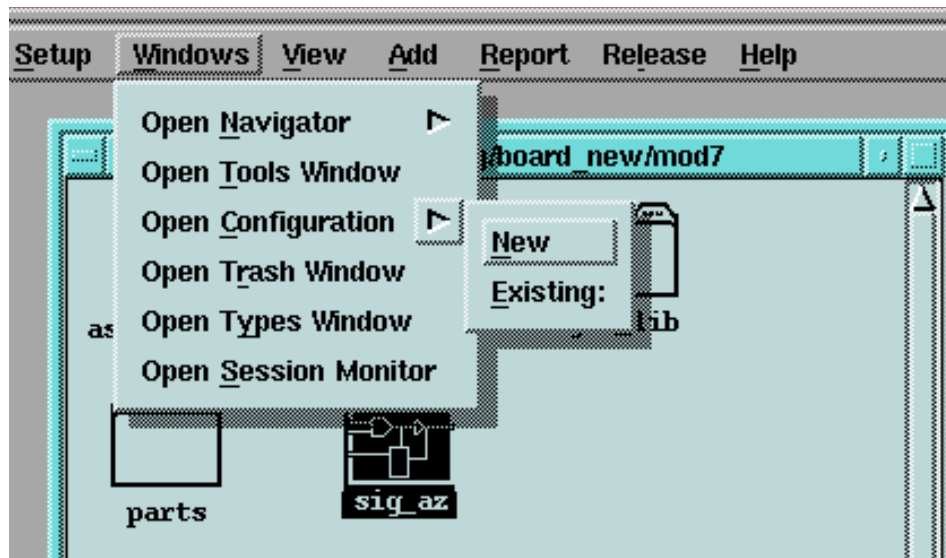
This adds the configuration object to the configuration for future use. If you do not save a configuration, you lose it when you close the Configuration window.

9. Close the Configuration window by choosing the **Window > Close** menu item.

## Creating a PCB-Only Configuration

In this part of the lab, you create a configuration that includes only PCB data.

1. In the Navigator window, move up the hierarchy so the contents of **mod7** are displayed, and choose the **Windows > Open Configuration > New** pulldown menu item.



2. Drag the ghost image of **sig\_az** from the Navigator window into the new Configuration window, activate the Configuration window, and click on the **Palette > Build** icon.

This creates a configuration based on default build rules. It is useful for finding which object pathnames to include or exclude.

3. Scan the configuration listing to identify which objects to specify in the Set Build Rules dialog box, select **sig\_az** in the Configuration window, and click on the **Palette > Set Build Rules** icon.
4. Complete the Set Build Rules dialog box as shown and click the **OK** button.

Set Build Rules			
Traverse Object Reference Hierarchy?		<input type="button" value="Yes"/>	<input type="button" value="No"/>
Filter Out MGC V7.X Design?		<input type="button" value="Yes"/>	<input type="button" value="No"/>
FILTERS			
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/sig_az/pcb/"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/sig_az/pcb_parts/"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/sig_az/design_"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/sig_az/comps_file"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/sig_az/grid_file"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/sig_az/gates_file"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/sig_az/geoms_file"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/parts/"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
Object Path:	<input type="text" value="/ user/jean/training/board_new/mod7/gen_lib/"/>	<input type="button" value="Include"/>	<input type="button" value="Exclude"/>
<input type="button" value="OK"/> <input type="button" value="Reset"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>			



- In the Configuration window, select all the top-level components in **sig\_az** (**DATA\_IO**, **BUF\_DRIVERS**, **MCM**, and so on) and any other schematic files. Click on the **Palette > Remove Entry** icon.

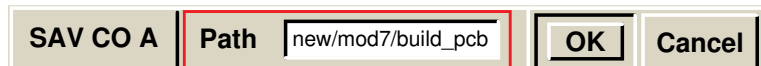
This removes most schematic information from the configuration.



6. Click on the **Palette > Build** icon.

To refine configuration you have just built, set the build rules to exclude more items. Each time you set build rules, build the configuration again to see the results.

7. Save the configuration by choosing [**Configuration**] **Save As**. Specify a pathname and **OK** the dialog box.



8. Close the Configuration window.

## Creating a Schematic Configuration

You can create a configuration to release only schematic information. This is useful for design experimentation, or for transfer to another group for reuse. Perform the following steps to create a schematic configuration of the DATA\_IO component:

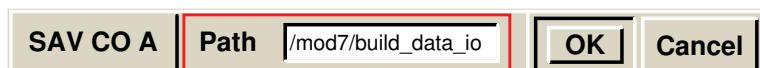
1. Choose **Windows > Open Configuration > New**.
2. In a Navigator window, display the contents of **sig\_az** and select and drag the ghost image of the **DATA\_IO** icon into the Configuration window.



3. Select **DATA\_IO** in the Configuration window, click on the **Palette > Set Build Rules** icon, and click on **Traverse Object Reference Hierarchy? No**.

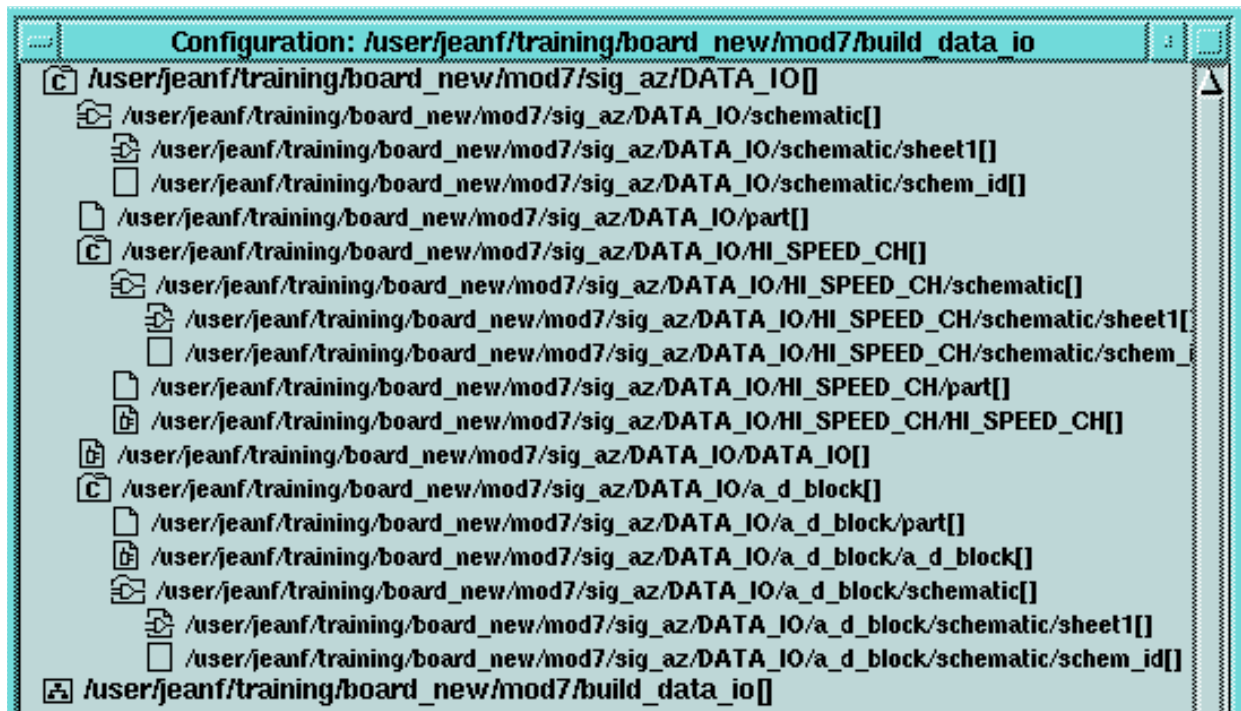
By building a configuration that does not traverse the reference hierarchy, you assume that whoever will use it already has libraries with all the necessary components.

4. Click on the **Palette > Build** icon.
5. Choose **[Configuration] Save As**, specify a name for the configuration, and **OK** the dialog box.



The pathname you specified is added to the configuration object, and the window banner displays the pathname.





6. Close the Configuration window by choosing **Close** from the menu in the upper-left corner of the window.

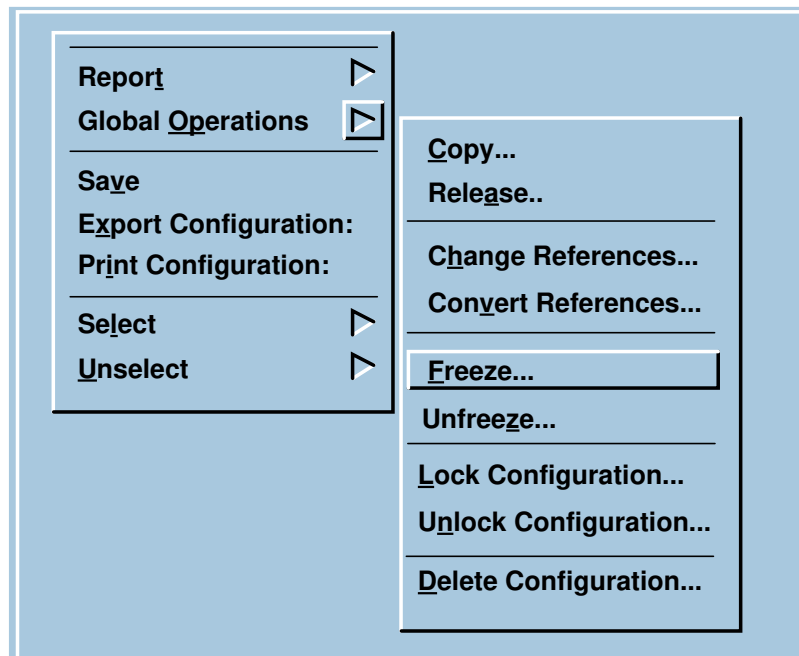
## Releasing a Design Configuration

In this section, you release the DATA\_IO configuration you created in the previous section.

1. In a Navigator window, show the contents of the directory where you previously wrote the configuration object.  
(...mod7/build\_data\_io.)
2. Click the Select mouse button on the configuration object, then choose the **[Navigator] Open > Configuration Window** popup menu item.



3. Lock the configuration by choosing the **[Configuration] Global Operations > Lock Configuration** popup menu item.



This prevents anyone else from modifying or deleting configuration entries while you perform the release.

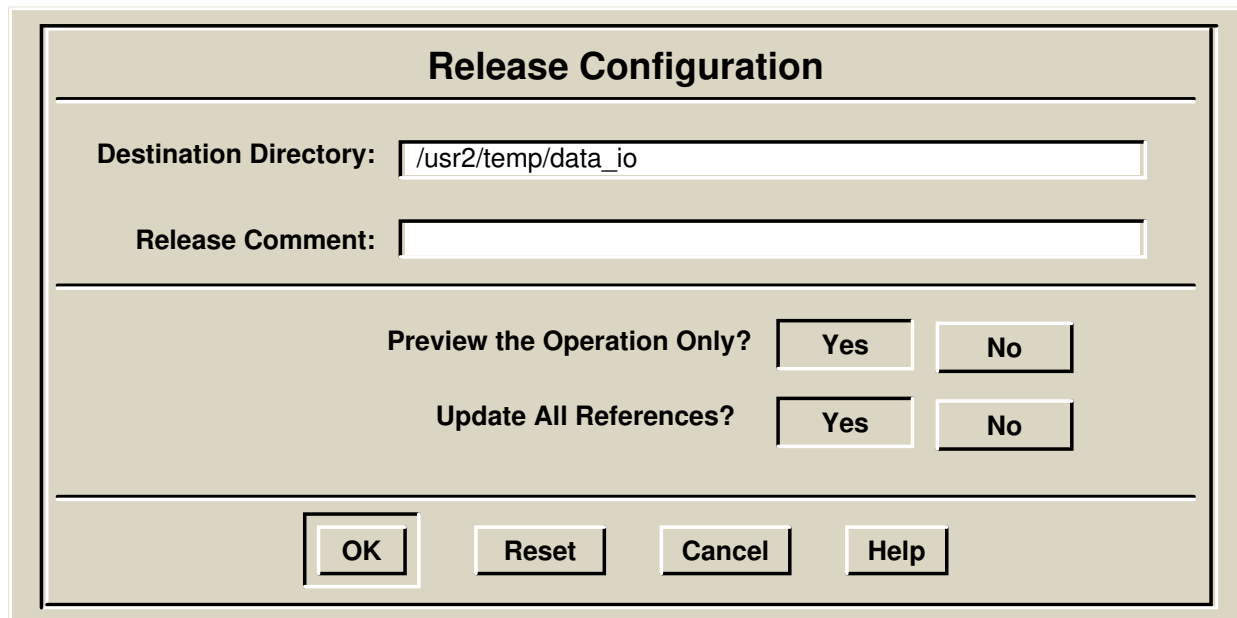


4. Click the **Palette > Build** icon to update the display to exactly reflect the contents of the configuration.

This step is recommended in case someone moved or deleted a configuration entry. Because you locked the configuration first, all secondary entries are locked as they are added to the configuration.



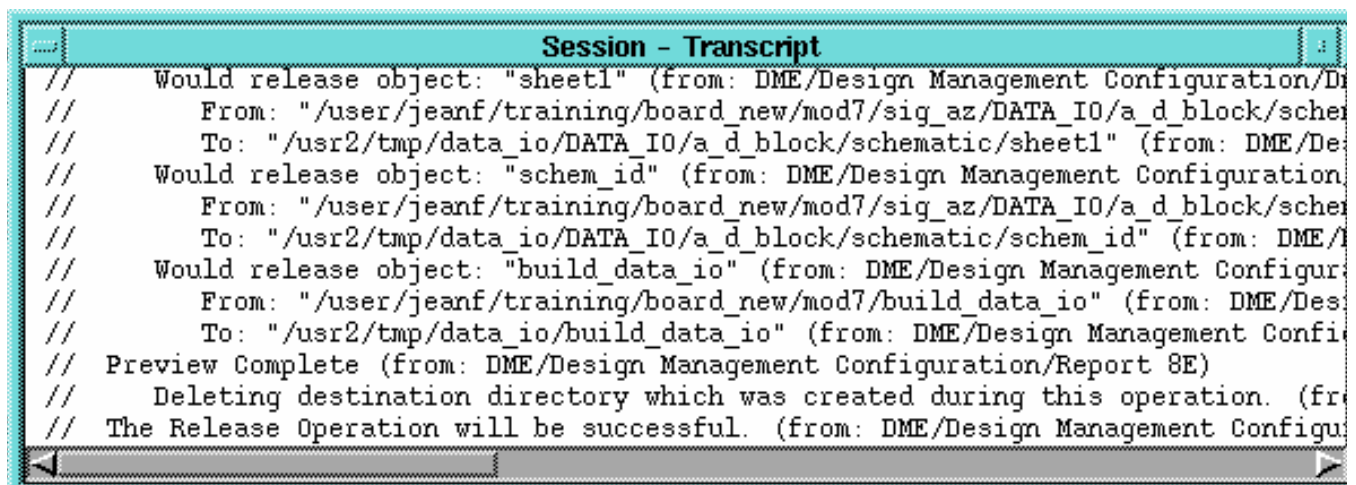
5. Click on the **Palette > Release** icon.
6. Enter a destination pathname in the Release Configuration dialog box, click on **Yes** for both **Preview** and **Update**, and **OK** the dialog box.



The image shows a 'Release Configuration' dialog box. It has a title bar with the text 'Release Configuration'. Inside, there are two input fields: 'Destination Directory:' with the value '/usr2/tmp/data\_io' and 'Release Comment:' which is empty. Below these fields are two groups of buttons. The first group is labeled 'Preview the Operation Only?' and contains 'Yes' and 'No' buttons. The second group is labeled 'Update All References?' and also contains 'Yes' and 'No' buttons. At the bottom of the dialog are four buttons: 'OK', 'Reset', 'Cancel', and 'Help'.

If the destination directory does not exist, a confirmation dialog appears when you execute the Release Configuration dialog box. It asks if you want to create the directory. Click on **Yes**.

Design Manager checks for naming conflicts in the destination directory, records the source and destination pathnames of all objects in the configuration, then deletes the temporary directory and issues a message saying whether the release would be successful.

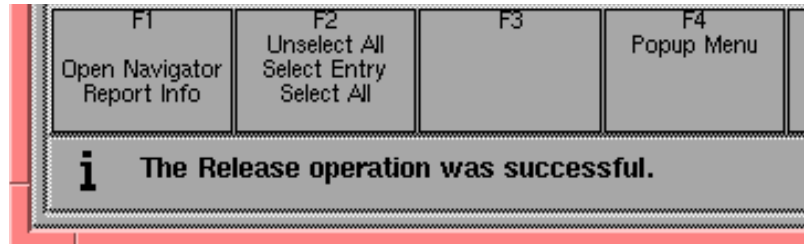


The image shows a 'Session - Transcript' window with a list of log messages. The messages describe the release process for three objects: 'sheet1', 'schem\_id', and 'build\_data\_io'. Each object's release is detailed with its source and destination paths. The process concludes with a 'Preview Complete' message, a note about deleting the destination directory, and a final confirmation that the release operation will be successful.

```
// Would release object: "sheet1" (from: DME/Design Management Configuration/D
// From: "/user/jeanf/training/board_new/mod7/sig_az/DATA_IO/a_d_block/sche
// To: "/usr2/tmp/data_io/DATA_IO/a_d_block/schematic/sheet1" (from: DME/De
// Would release object: "schem_id" (from: DME/Design Management Configuration
// From: "/user/jeanf/training/board_new/mod7/sig_az/DATA_IO/a_d_block/sche
// To: "/usr2/tmp/data_io/DATA_IO/a_d_block/schematic/schem_id" (from: DME/
// Would release object: "build_data_io" (from: DME/Design Management Configur
// From: "/user/jeanf/training/board_new/mod7/build_data_io" (from: DME/Des
// To: "/usr2/tmp/data_io/build_data_io" (from: DME/Design Management Conf
// Preview Complete (from: DME/Design Management Configuration/Report 8E)
// Deleting destination directory which was created during this operation. (fr
// The Release Operation will be successful. (from: DME/Design Management Configur
```

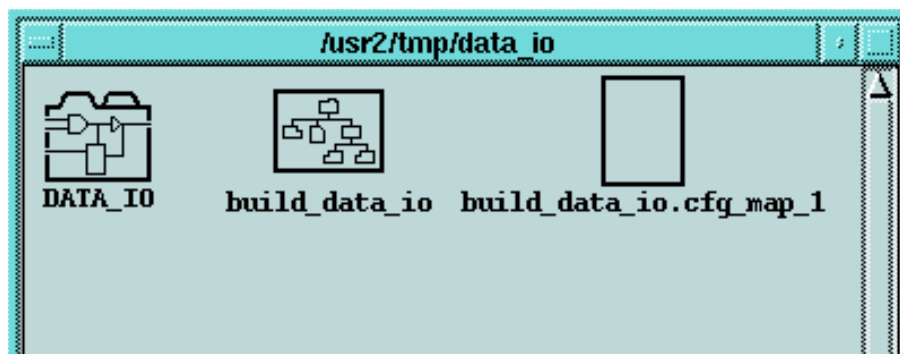
7. Close the Transcript window, then click the **Palette > Release** icon. In the dialog box, specify **No** for **Preview** and **OK** the dialog box.

Design Manager processes all configuration entries, as in the preview, then updates references and issues a message when done.



8. In a Navigator window, navigate to the destination directory you specified for the release.

The destination directory into which you released DATA\_IO contains the *DATA\_IO* directory, the configuration object, and a location map.



## Archiving and Restoring a Design

Usually, a system administrator uses the UNIX **tar** command to archive and restore data, and most network users need not be concerned about how to do it. In this part of the lab, you use **tar** to archive your released directory, then you restore the directory to another location and fix design references.

### Archiving

1. In an operating system shell, change your working directory to the one that contains the released data:

```
cd /usr2/tmp
```

2. Invoke **tar** on the directory:

```
tar cf data_io.tar data_io
```

Because most design data is large, it is good to **tar** the data before moving or copying it to another location.

3. Create a directory and copy the new file into it:

```
mkdir training
```

```
cp data_io.tar training/data_io.tar
```

The data is now archived. The target directory could have been a tape or other storage area.

### Restoring

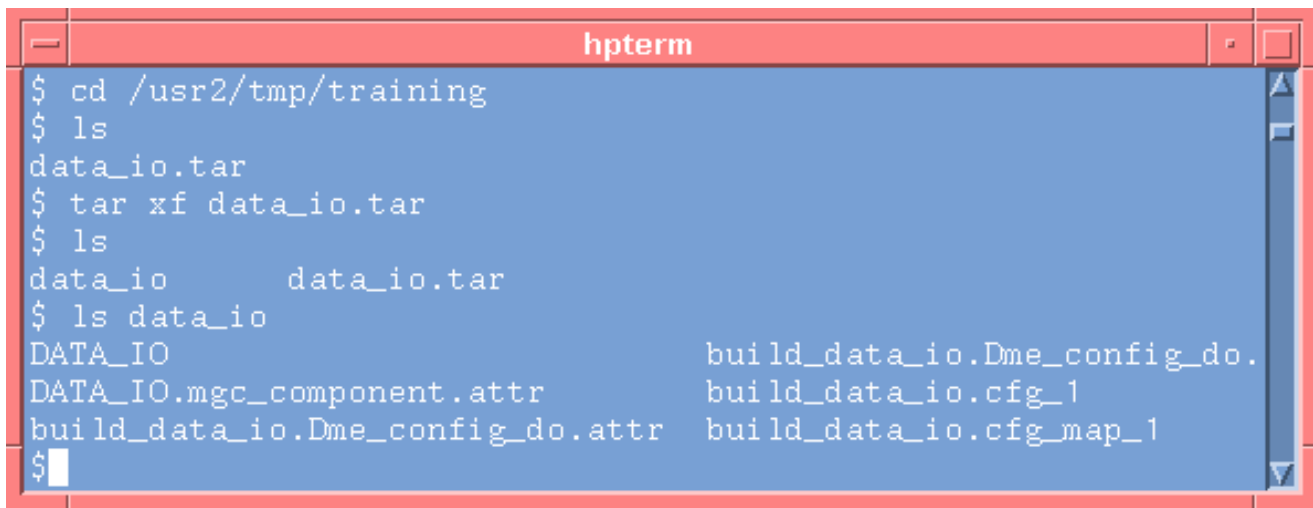
Now you restore your data in its new location.

1. Move to the directory containing your data, and restore the directory structure:

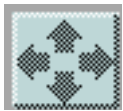
```
cd /usr2/tmp/training
```

```
tar xf data_io.tar
```

The following window shows the transcript with a directory listing.



```
hpterm
$ cd /usr2/tmp/training
$ ls
data_io.tar
$ tar xf data_io.tar
$ ls
data_io      data_io.tar
$ ls data_io
DATA_IO      build_data_io.Dme_config_do.
DATA_IO.mgc_component.attr  build_data_io.cfg_1
build_data_io.Dme_config_do.attr  build_data_io.cfg_map_1
$
```



2. In a Design Manager Navigator window, click on the **Goto** button and navigate to `/usr2/tmp/training/data_io`.



3. Select **DATA\_IO** by clicking on the icon in the Navigator window, then click on the **Palette > Check Refs** icon.

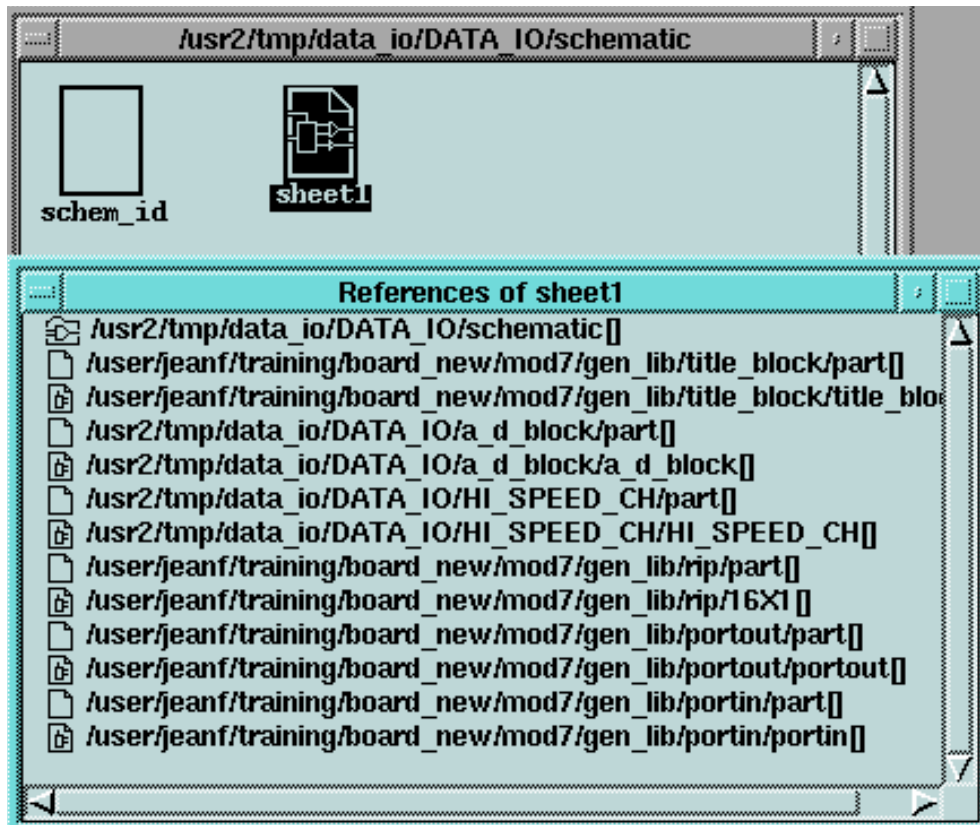
Design Architect checks the references in DATA\_IO and reports that no broken references were found. This is because, in this case, all referenced objects exist, even though they probably would not in a real work environment.

4. Use the navigator to display the contents of:

*/usr2/tmp/training/data\_io/DATA\_IO/schematic*

5. Click on the **sheet1** icon to select it, then choose the **[Navigator] Report > Show References** popup menu item.

Design Manager displays the reference pathnames for objects on sheet1.

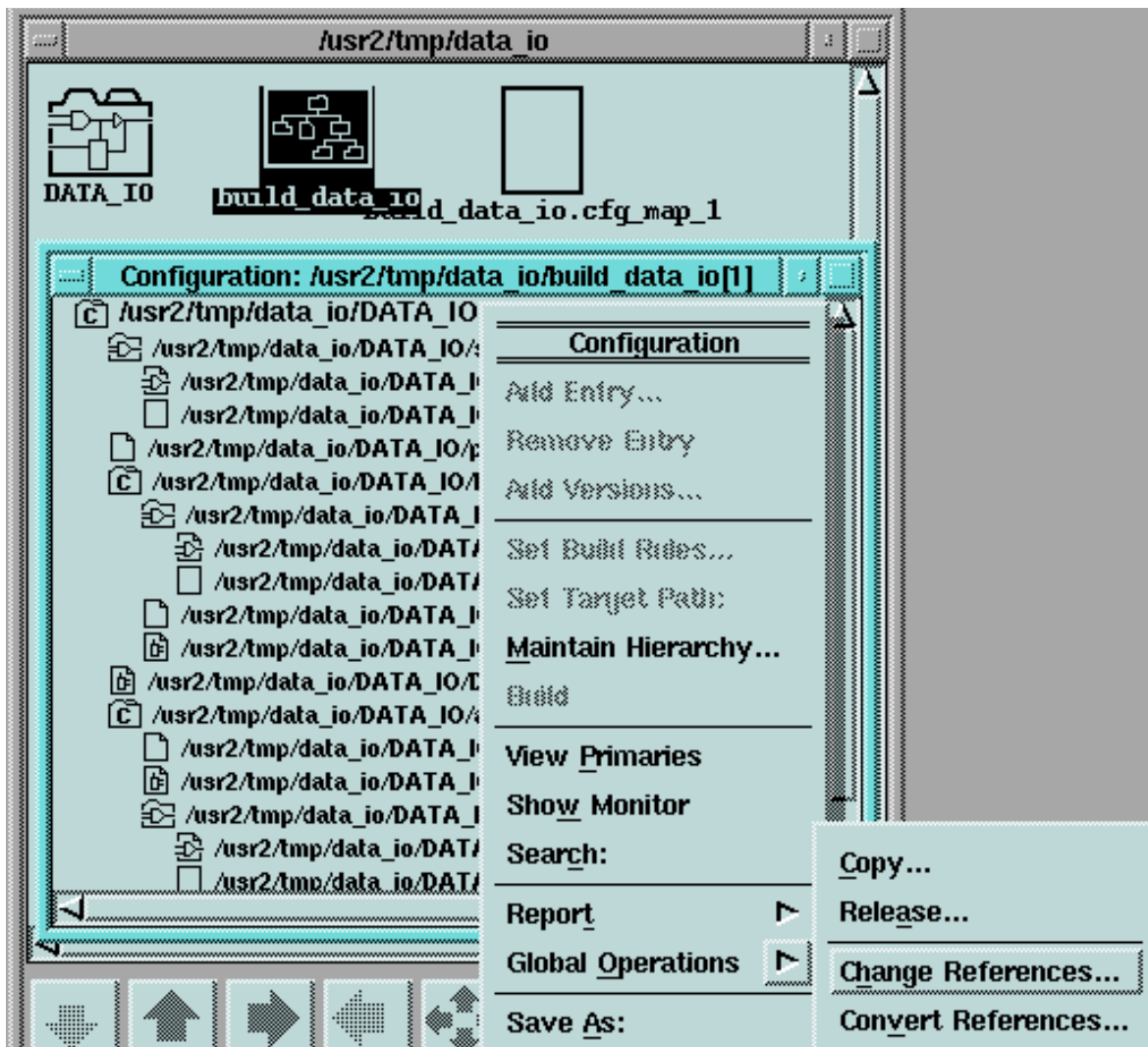


6. Move the References window to the side of the client area so it will not be covered by a dialog box, then navigate back up to the */usr2/tmp/training/data\_io* directory and click on **build\_data\_io**.
7. Choose the [Navigator] Open > Configuration Window popup menu item.

A Configuration window displays a copy of the original configuration object that was archived with the configuration.

8. In the Configuration window, choose the [Configuration] Global Operations > Change References popup menu item.

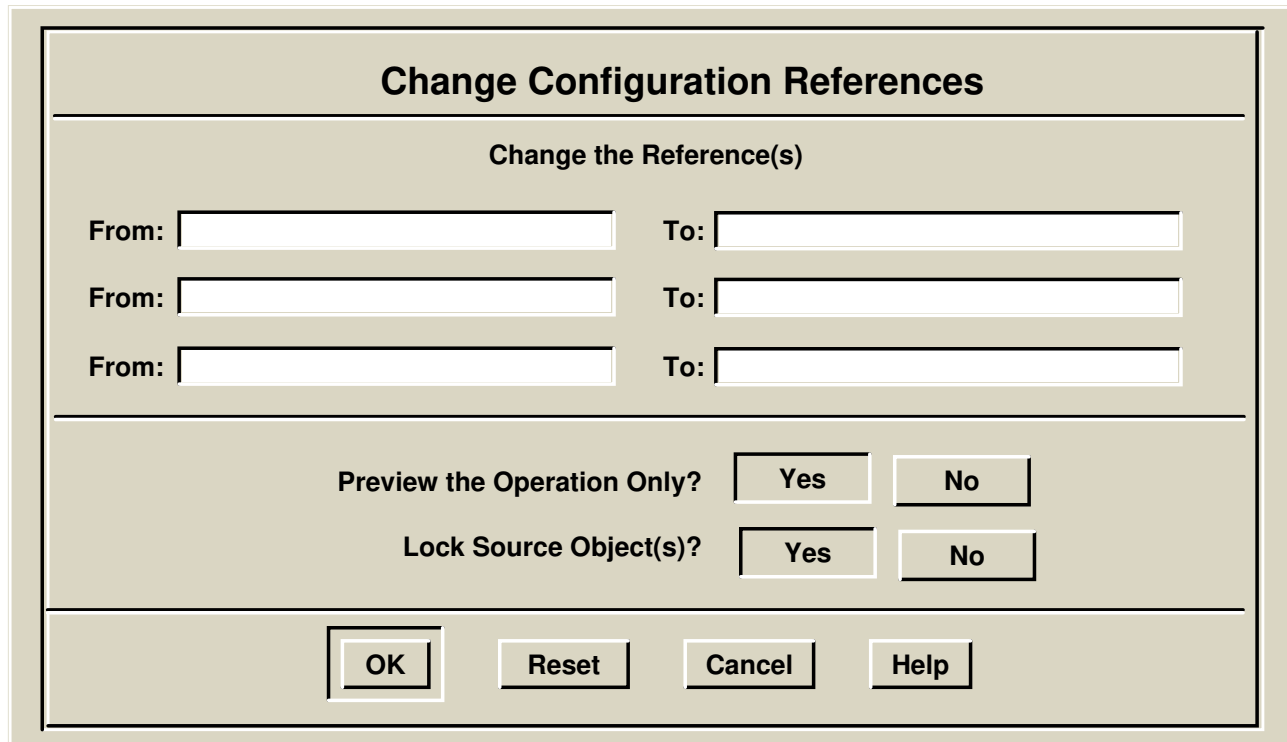




9. The Change Configuration References dialog box displays for you to specify replacement reference pathnames. You can use the References window to see the existing pathnames that you need to change.

10. **Preview** the change before actually changing the references.

In the following figure, no replacement pathname is shown for the *parts* and *gen\_lib* references. If you have a new location for these libraries, then enter that new pathname.



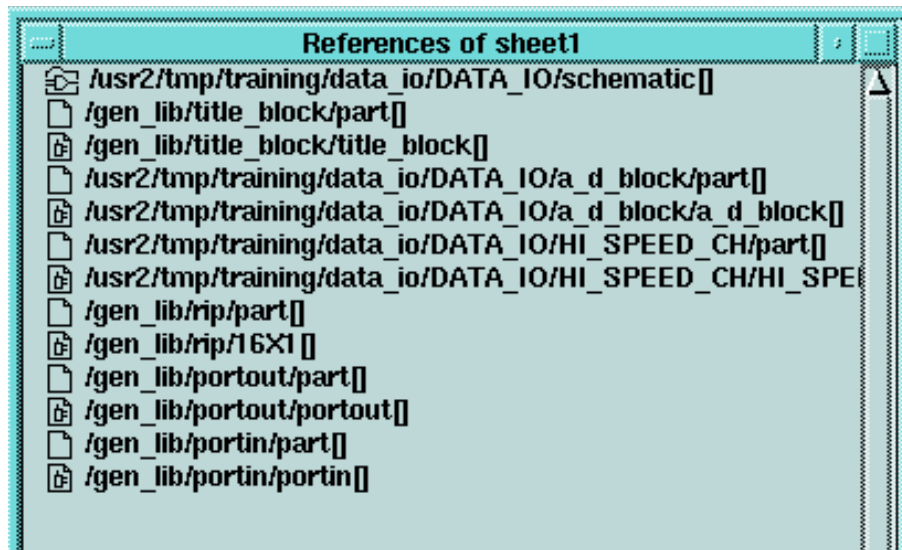
The dialog box is titled "Change Configuration References". It contains a section "Change the Reference(s)" with three rows of "From:" and "To:" text input fields. Below this, there are two rows of checkboxes: "Preview the Operation Only?" and "Lock Source Object(s)?", each with "Yes" and "No" buttons. At the bottom, there are four buttons: "OK", "Reset", "Cancel", and "Help".

Design Manager checks all the potential reference changes, then displays a message.



11. If no problems are found, repeat the step, specifying **No Preview**.
12. Navigate to `/usr2/tmp/training/data_io/DATA_IO/schematic`, click on the **sheet1** icon to select it, then choose the **[Navigator] Report > Show References** popup menu item.

This shows the results of changing references. Notice that Design Manager removed the pathname before *gen\_lib*, exactly as specified; if the **To:** text entry field had been filled, the pathname would have been replaced.



13. If you have enough lab time, use the Design Manager to copy the *gen\_lib* and *parts* libraries to another location, then change these references in the design.

Congratulations! You have completed the *Board Station for New User's Training Series*.

## Removing Training Library Links from `pcb_parts` Directory

This is an optional procedure. If you want to keep the training data so that you can refer to it in the future, you should not complete this procedure. If you want to restore your `pcb_parts` directory in your account to the state it was in before you copied the training, you should complete this procedure. This procedure does not delete all your training data, it removes only the training library links from your `pcb_parts/user_geom` directory.



*If you plan to complete Module 8: Managing Design Changes, skip this portion of the lab exercise. You need these links for the lab exercises in Module 8.*

1. Change your directory to the `user_geom` directory in your `pcb_parts` directory by executing the following in a shell:
2. Delete the training links you created by executing the following in a shell:

```
cd your_path/pcb_parts/user_geom  
  
rm mgc.trng.drawings  
rm mgc.trng.components  
rm mgc.trng.hardware  
rm mgc.trng.padstacks  
rm mgc.trng
```

The training links are removed.

# INDEX

## A

- Add
  - board side view 5-8
  - dimensions 5-13
- Add Artwork Layer dialog box 2-6
- Aperture table
  - creating report 5-16, 5-17
- Archiving a configuration 6-33
- Archiving designs 6-32
- Artwork
  - simulating data 2-27
- Artwork > Change Aperture Table > Change Aperture Table 2-35
- Artwork > Change Aperture Table > Change Power Aperture 2-36
- Artwork > Change Aperture Table > Fill Aperture Table 2-34
- Artwork > Create Artwork Data 2-48
- Artwork > Simulate Artwork Data 2-49

## B

- Bills of material
  - customize 5-18
- Board
  - side view, add 5-8
- Build configuration 6-17
- Build rules
  - filters 6-15
  - object path 6-15
  - object type 6-15
  - property 6-16

## C

- Change
  - reference designators 1-3
- Change All References Automatically dialog box 1-6
- Change Drill Table dialog box 4-3
- Change Geometry > Add Artwork Void Outline 3-24
- Change Milling Tool dialog box 4-9
- Change Reference > Change All Reference Names Auto 1-13
- Change Reference > Change Reference > Change Reference Name 1-12
- Change Reference > Change Selected Reference Names Auto 1-13
- Change Reference dialog box 1-4
- Check > Aperture Table 2-47, 2-48
- Configuration 6-8
  - archiving 6-33
  - building 6-17
  - copying 6-26
  - deleting 6-31
  - freezing 6-30
  - locking 6-19
  - object path 6-15
  - object type 6-15
  - releasing 6-19
  - restoring 6-33
  - saving 6-18
  - secondary entry 6-17
  - updating references 6-34
  - versions 6-28
  - viewing containment 6-23
  - viewing design hierarchy 6-25
  - viewing primary entries 6-26
- Copying 6-26
- Create Artwork Data dialog box 3-12
- Create Drill Data dialog box 4-4
- Create Manufacturing Panel dialog box 3-5

## INDEX [continued]

### Customize

- bills of material 5-18
- drill schedules 5-5

## D

### Data

- simulating artwork 2-27
- viewing drill and mill 4-14

### Deleting a configuration 6-31

### Design configuration 6-8

### Design objects

- archiving 6-32
- freezing 6-28
- restoring 6-32
- updating references 6-34
- versions 6-28

### Dialog boxes

- Add Artwork Layer 2-6
- Change All References Automatically 1-6
- Change Configuration References 6-34
- Change Drill Table 4-3
- Change Milling Tool 4-9
- Change Reference 1-4
- Copy Configuration 6-27
- Create Artwork Data 3-12
- Create Drill Data 4-4
- Create Manufacturing Panel 3-5
- Release Configuration 6-21
- Set Grid 3-7

### Dimensions

- adding 5-13
- examples 5-10
- ordinate type 5-15

### Drawing > Add Board 5-39

### Drawing > Add Board Side View 5-47

### Drawing > Add Format Block > Add Drill Schedule 5-45

### Drawing > Add Pointer 5-25

### Drawing > Add Thieving Pattern > Add Pattern

3-26

### Drill > Change Drill Table > Change Drill Table 5-43, 5-44

### Drill > Change Drill Table > Fill Drill Table 4-18

### Drill > Create Drill Data 4-19

### Drill data

- create 4-4
- view or write layers 4-14

### Drill schedules

- customize 5-5

### Drill symbols

- assign to drill sizes 4-7, 5-17

### Drill table

- create 4-3, 4-4

### Drill table report 5-17

## E

### Examples

- dimensioning 5-10

## F

### Fonts, text 5-11

### Freezing objects 6-28

## G

### Geometries > Create Geometry > Drawing 5-38

### Geometries > Create Geometry > Manufacturing Panel 3-19

### Geometries > Create Geometry > Test Coupons 3-18

### Gerber

- simulate artwork data 2-27

## INDEX [continued]

### H

Hole symbols  
    assign to drill sizes 4-7, 5-17

### L

Layers  
    viewing drill and mill data 4-14

### M

Manufacturing reports 5-16  
Mill table report 5-17  
Milling > Change Tool 4-20  
Milling > Change Tool Size 4-21  
Milling > Start Tool Path 4-20  
Milling data  
    view or write layers 4-14  
Milling tool  
    change size 4-12

### O

Object  
    finding type 6-16  
Object information report window 6-16  
Object type 6-16

### P

Palette > Build 6-17  
Palette > Copy 6-26  
Palette > Release 6-20  
Palette > Report Info 6-15  
Photoplotting  
    simulate artwork data 2-27

### R

Reference designators  
    changing 1-3  
References  
    updating 6-22  
Release 6-19  
    comment 6-21  
    destination 6-21  
    preview 6-21  
    versioned objects 6-22  
Release PCB Data  
    pcb container only 6-5  
Release PCB data  
    entire design 6-3  
    Release PCB 6-3  
    schematic only 6-6  
Releasing a Design 6-1  
Report > Attributes > Any Geometry 2-33  
Report > Bill of Materials 5-49  
Report > Drill Table 4-18  
Report > Object Info 6-15  
Reports  
    aperture table 5-16, 5-17  
    drill and mill tables 5-17  
    manufacturing 5-16  
Restoring a configuration 6-33

### S

Save configuration 6-18  
Set Grid dialog box 3-7  
Shapes > Extended Menu > Add Board Layer  
    Block 3-28  
Simulate  
    artwork data 2-27  
Sizes, drill  
    assign to drill symbols 4-7, 5-17  
Symbols, drill  
    assign to drill sizes 4-7, 5-17

## INDEX [continued]

### T

Text 5-11  
Text > Add Text File 5-42  
Tool size, milling  
    change size 4-12

### V

View > Drill Symbols On 5-45  
Views  
    configuration primaries 6-26  
    containment hierarchy 6-23  
    primary and secondary entries 6-25  
    side, add for board 5-8



# 欢 迎 光 临

**微波 EDA**

<http://www.mweda.com>

**EDA学习网**

<http://edastudy.ik8.com>

<http://www.mweda.com>

<http://edastudy.ik8.com>

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

## 破解软件类：微波仿真软件

ADS2004A	ADS2003C	ADS2003A		
HFSS9.2	HFSS9.1	HFSS9.0	HFSS8.0	
Ansoft Designer1.1	Ansoft Serenade8.71	Ansoft Maxwell 10	Ansoft SIWave	
Microwave Office2002	Sonnet Suite Pro 9.52	CST5.0		
Super NEC 2.5	Zeland IE3D9.2	XFDTD 6.0		

## 破解软件类：PCB 工具软件

Mentor EN2004	Mentor EN2002	Mentor ePD2004	Mentor SDD2004
Mentor WG2004	Mentor WG2002	Mentor ISD2004	
PowerPCB5.0	PowerLogic5.0	Orcad10.3	
PADS2005	PADS2004		
Cadence SPB15.2 (Allegro 15.2)		Cadence PSD15.0 (Allegro 15.0)	

## 软件学习、培训教程：

**Mentor EN:** Mentor EN 原版培训教程  
Mentor EN 视频教程

**Mentor WG:** Mentor Expedition / Mentor WG 中文用户手册  
Mentor Expedition ( WG ) PCB Training Workbook  
Mentor DxDesigner Design Processing Training Workbook

**HFSS :** HFSS 9.2 入门与提高教程  
HFSS 9.0 入门与提高教程  
HFSS9 视频教程  
HFSS9 Training Tutorials  
HFSS8 Training Manual  
电子科大 HFSS8 教程

**ADS:** ADS 中文基础教程  
ADS 设计实验教程  
ADS 入门教程  
ADS2003 Training Workbook: ADS2003 fundamental  
ADS Training Workbook for Momentum  
ADS Customization Training Workbook

Using ADS Communication Systems Designer

Using ADS to Design WCDMA/3GPP Communication Systems

**Ansoft Designer:**

Ansoft Designer 2.1 Full Book

Ansoft Designer System Training Guide

Ansoft Designer 1.1 入门与提高

Ansoft Designer 多媒体教程

Ansoft Designer Training 讲义

**Cadence Allegro:**

Allegro 15.2 原版教程

Concept-HDL 15.2 原版教程

Allegro 14.2 原版教程

Allegro 视频教程

Allegro PCB Layout 高速电路板设计

注：本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D , Zeland IE3D, PowerPCB , Orcad等的培训教程。以及射频/微波/天线类英文原版书籍。详情可登陆：

<http://www.mweda.com> 或 <http://edastudy.ik8.com>